

**Pro/ENGINEER<sup>®</sup> 2001**

**Pro/ASSEMBLY<sup>™</sup> (Basic)  
Topic Collection**

**Parametric Technology Corporation**

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# About Assembly Mode Functionality

Just as you can combine features into parts, you can also combine parts into assemblies.

Assembly mode in Pro/ENGINEER enables you to place component parts and subassemblies together to form assemblies, as well as to design parts based on how they should *fit together*. You can then modify, analyze, or reorient the resulting assemblies.

## Assembly Functions

Pro/ENGINEER provides basic assembly tools, and various Pro/ENGINEER modules give you additional functionality for assembly operations.

Pro/ASSEMBLY supports the design and management of large and complex assemblies through the use of powerful tools such as simplified representations, interchange assemblies, and the Design Manager.

## Simplified Representations

Simplified representations are variations of a model you can use to change the view of a particular design, enabling you to control which members of an assembly Pro/ENGINEER brings into session and displays. This lets you tailor your work environment to include only the information of current interest to you. You can, for example, temporarily remove a complicated subassembly from memory that is unrelated to the portion of the assembly on which you need to work. You can also substitute a complicated subassembly or part with a simpler part or envelope.

Using advanced performance tools, you can speed up the retrieval process and general work performance of large assemblies using simplified representations.

## Interchange Assemblies

An interchange assembly is a special kind of assembly that you can create and then use in a design assembly. An interchange assembly consists of models that are related either by function or representation. You can create both functional interchanges (to replace functionally equivalent components) and simplify interchanges (to substitute components in a simplified representation) in the same interchange assembly. Interchange assemblies, like family tables and layouts, provide a powerful method of automatic replacement.

Online Help documentation for Advanced Pro/ASSEMBLY Extension provides detailed information. Foundation users can use interchange assemblies; however, an Advanced Pro/ASSEMBLY Extension license is required for creating them.

## Skeleton Models

The skeleton model of the assembly is the framework of the assembly. A skeleton model is a specialized component of an assembly that defines skeletal, space claim, interface, and other physical properties of an assembly design that you can use to define geometry of components. In addition, you can use skeleton models to perform motion analysis on an assembly by creating placement references to the skeleton model and then modifying the skeleton dimensions to imitate motion.

Skeleton models can be used to capture in a central location design criteria defined in the subassembly or delivered from a higher-level assembly. Using skeleton models in more than one assembly allows you to distribute design criteria associatively throughout the product structure. When design criteria change, updating is propagated to affected components. Skeleton models provide a clearly understood hierarchy of driving design criteria, they provide an organized display, and they allow improved performance. Skeleton models are the recommended mechanism for controlling top-level design iterations, and you can use them to facilitate task distribution.

Skeletons are represented by a unique icon in the Model Tree because their functional characteristics are significantly different from those of other components. Skeleton models can be filtered out the BOM and drawing views and can be specially handled during the creation and manipulation of simplified representations

and Shrinkwrap features. Skeleton models are placed before all other components with solid geometry in the model tree. Reference scope control settings can be used to restrict making assembly placement references to skeleton models only.

Skeleton models, like regular components, can be replaced by both family table instances and other skeleton models. You can copy a part model component into a new skeleton model, as long as the part model satisfies the skeleton model criteria. You can generate a native skeleton model, based on a native part model, and have it replace the part model in an assembly, with all references remapped to the new skeleton model. This effectively allows a part to be designated as a native skeleton model, through the use of a new model file.

Starting with Release 2001, skeleton models can maintain their own family tables. This enhancement allows assemblies to maintain different skeleton instances across a family table.

Although skeletons can be created only within an assembly, they can be retrieved, operated upon, and saved as ordinary parts.

Foundation users can use and modify skeleton models; however, an Advanced Pro/ASSEMBLY Extension license is required for creating them.

## The Design Manager

Within the Pro/ASSEMBLY module, the Design Manager functionality provides top-down design tools, reference control and investigation tools, and advanced performance tools.

- Using top-down design tools, you can set up a well-structured design containing skeleton models and copied geometric and datum references.
- Using reference control and investigation tools, you can view and manage the complex web of dependencies that evolve with the design. These tools enable you to easily trace and understand the references that you make among features in a design. They clarify the external reference relationships that exist among models in an assembly.

## Pro/NOTEBOOK

The optional Pro/NOTEBOOK module supports top-down assembly design with tools that enable you to create hierarchically-linked assembly layouts. Online Help documentation for Advanced Pro/ASSEMBLY Extension provides detailed information about layouts.

## Pro/PROCESS for ASSEMBLIES

The optional Pro/PROCESS for ASSEMBLIES module enables you to create explode states in assemblies to define the exploded position of all components.

## Working with Assemblies

To work with an assembly, use the **File** menu to open or create an assembly file. The ASSEMBLY menu displays the following options:

- **Component**—Manipulates assembly components (using the COMPONENT menu).
- **Feature**—Manipulates assembly features (using the ASSY FEAT menu).
- **Modify**—Modifies assembly or component dimensions and features (using the ASSEM MOD and MODIFY menus).
- **Restructure**—Modifies assembly groupings, moving components from one assembly or subassembly to another (using the RESTRUCTURE menu).
- **Mechanism**—Allows you to study the allowable motion of the assembly using Mechanism Design Extension (MDX).
- **Simplfd Rep**—Creates, modifies, or sets a simplified representation (using the SIMPLFD REP menu).
- **Design Mgr**—Accesses tools to manage assembly design (using the DESIGN MGR menu).
- **ExplodeState**—Creates, sets, and modifies explode states of an assembly (using the EXPLD STATE menu).
- **Regenerate**—Updates modified part and assembly dimensions (using the PRT TO REGEN menu).

- **Relations**—Edits parametric labels and adds or edits constraint equations (using the MODEL REL and RELATIONS menus).
- **Family Tab**—Edits assembly family tables or creates assembly instances (using the Family Table dialog box).
- **Set Up**—Assigns assembly mass properties, and specifies length units, mass units, dimension bounds, and other set up properties (using the ASSEM SETUP menu).
- **Program**—Provides an option (Pro/PROGRAM) to create a program to control the design of parts in an assembly (using the PROGRAM menu).
- **Integrate**—Retrieves integration project files (created in Pro/PDM) and generates difference reports to resolve differences between source and target assemblies (using the INTEGRATE menu).
- **Copy From**—Copies entire assemblies or subassemblies into the new assembly.

## To Copy an Assembly

Using **Copy From** from the ASSEMBLY menu, you can copy an entire assembly or subassembly into a new assembly.

You can use **Copy From** to copy features but not parts or any solid geometry. **Copy From** is available only for copying an assembly that is empty (but that can contain assembly features) into an assembly that is completely empty (one that does not contain even assembly features).

Recommended practice is to use start components and default templates. This functionality may often be used more efficiently than **Copy From** to replicate assemblies. Online Help documentation for Core applications provides detailed information about using default templates.

## Using Start Components and Default Templates

A *start component* is a standard component that you can copy to create new parts or assemblies. Start components can contain relations, layers, views, and parameters subject to the following conditions:

- A start part should not have any external dependencies.
- A start assembly should contain only assembly features.

**Note:** Start assemblies, which you can use to create a new subassembly, can contain assembly-level data (such as features, parameters, and layers) but cannot contain components or skeleton models.

If the selected start part or assembly has a family table, you will be prompted to select a particular instance of that family table, and the information that is associated with that instance will be copied to the newly created component. If you select the generic instance, the family table will *not* be copied over to the new component.

If you have specified a default template (you can set the configuration file option `start_model_dir` to specify its location), the system uses that template, or start model, for the part. Using a template as a start model allows you to include critical layers, datum features, and views in the model.

Online Help documentation for Core applications provides detailed information.

## Opening Simplified Representations by Default

You can customize the system to display a prompt allowing you to select a simplified representation to retrieve, or to automatically open a specific simplified rep.

With the configuration file option `open_simplified_rep_by_default` set to `yes` (the default is `no`), whenever an appropriate object is opened, the system automatically opens the Open Rep dialog box, prompting you to select a simplified representation to retrieve.

In the configuration file, you can also preselect the name of an assembly simplified representation by entering:

```
open_simplified_rep_by_default yes <rep name>
```

If the representation exists, the name is highlighted in the Open Rep dialog box, and you click Open. If the representation does not exist, you are prompted to create a new simplified representation with that name. Note that the default representation name applies only to assembly simplified reps in Pro/ENGINEER 2001.

Working with simplified representations is recommended best practice, and presenting the Open Rep dialog box helps you avoid opening large objects in their Master Representations by mistake, thereby tying up significant amounts of time unnecessarily.

## Regenerating to Update Modified Parts

You must use the **Regenerate** command to update the modified parts after you make dimensional modifications. You can select individual parts to regenerate; however, the parts chosen this way will be regenerated in the order in which they were chosen.

## Saving Undisplayed Objects

You can use a configuration file option to protect yourself from erasing modified, unsaved objects that are not currently displayed.

You can safely close a window containing a modified part without losing your work, and the system therefore does not issue a warning. However, when you erase all objects currently not displayed, you may inadvertently lose objects that you have modified but not yet saved.

Set the configuration file option `prompt_on_erase_not_disp` to specify whether to display a prompt allowing you to save your modified and unsaved objects. When set to `yes`, when you choose **File > Erase Not Disp**, the system provides a prompt for each modified, unsaved object, allowing you to save the object before it is erased. When set to `no` (the default), the system immediately erases all undisplayed objects.

## Using Save A Copy with Assemblies

When you save a copy of an assembly, you have the option to copy any or all members of the assembly. However, you *must* copy and rename any components that have external references.

Click **File > Save Copy As**, then enter a new name for your assembly and click **OK**. The **Save A Copy** dialog box opens. It provides a convenient way to provide new names for the desired assembly components. If a family table is used with your assembly, the **Table Driven** dialog box opens. You can save your assembly as an instance or new model.

The source assembly is shown in a tree-like structure so you can see its hierarchy of components. By default each component is selected to be reused. To provide a new name, the user must select components and specify "New Name".

For each component, you can choose to give it a new name or to reuse the component in the source subassembly. If a subassembly is chosen to be renamed, all components in that subassembly can also be renamed using the default new name or remain as "unused". If you change all components to "new name", change the subassembly to "reuse", and then back to "new name", the status of the components change automatically to "new name". If a subassembly is reused, then all components in that subassembly are reused.

When you copy an assembly, you can copy any or all of the components that make up the assembly.

In the Rule section of this dialog box, you can establish a rule for a renaming convention. The default renaming convention is the old name appended with a default suffix. You can change the suffix and apply it to selected components. You can also create a template for other renaming conventions including a suffix or prefix.

**Note:** Click the right mouse button in the tree-like section to view the pop-up menu. To select multiple items to

be resumed or renamed, hold down the CTRL key and then click components or subassemblies.

## Using the Assembly Model Tree

A graphical, hierarchical representation of the assembly is shown in the Model Tree window. The nodes of the Model Tree represent the subassemblies, parts and features that make up an assembly. Icons, or symbols, provide additional information. You can expand or compress the tree display by double-clicking with the left mouse button on the name of the component.

**Note:** The system displays only one Model Tree window at a time—the tree for the active model.

The Model Tree can be used as a selection tool, allowing objects to be quickly identified and selected for various component and feature operations. In addition, system-defined information columns can be used to display information about components and features in the Model Tree.

The Model Tree contains a pop-up menu, providing direct access to the following Assembly operations:

- Modify an assembly or any component in an assembly
- Open the component model
- Redefine component constraints
- Reroute, delete, suppress, resume, replace, and pattern components
- Create, assemble, or include a new component
- Create assembly features
- Create notes (Online Help documentation for Core applications provides detailed information)
- Control references
- Access model and component information
- Redefine the display status of all components
- Redefine the display status of individual components
- Fix the location of a packaged component
- Update a Shrinkwrap feature

**Note:** You can invoke an operation from the Model Tree window only when no other operation is active in the system.

## To Highlight the Selected Component

When you select a component, the current model that contains that component is not highlighted automatically, but you can choose **HiliteCurMdl** to highlight it.

You can also choose **Model Tree Setup** from the **View** menu, and **Highlight Model**.

## To Use the Pop-Up Menu in the Assembly Model Tree

1. In the Model Tree window, position the cursor over the name of the assembly, part, or subassembly on which you want to operate.  
Use the **Shift** or the **Ctrl** key with the right mouse button to select multiple components.
2. Click and hold the right mouse button. A menu appears in the Model Tree, and the component is highlighted in the assembly.
3. Move the cursor to the menu command you want.
4. Release the mouse button. The command is selected, and you can follow the standard procedures to

complete the process.

**Note:** You can invoke an operation from the Model Tree window only when no other operation is active in the system.

Online Help documentation for Core applications provides further information about the Model Tree.

## To Add Information Columns to the Model Tree

You can add and remove system-defined columns to the Model Tree to display different types of information about components and features currently displayed in the Model Tree. You can expand and collapse the Model Tree window, and the system updates the columns accordingly.

1. Choose **Tree > Column Display...** to add a column to the Model Tree. The **Model Tree Columns** dialog box opens. If the Model Tree is embedded, choose **View > Model Tree Setup > Column Display** to open this dialog box.
2. Click the **Type** list, and select the name of a column from the list of types of information.
3. Click the right arrow command button to add the specified column to the **Displayed** list of columns.
4. Click **OK** to close the dialog box. The new column now appears in the Model Tree window. The system displays appropriate information in this column.

The columns display the following:

- Information items
  - Status (regenerated, packaged, suppressed/suppression order, and so on)
  - Feature information (number, ID, type, name)
  - Reference control
  - Copied references
  - Model size
- Feature parameters
- Model parameters
- Simplified representations
- Visual (display) mode
- Layer information
- Notes

## Using Assembly Family Table Instances

You can create a family table of the assembly to represent the design optional deviations, or a family of similar assemblies.

All items that can be added to the part family table can also be added to the assembly family table. In addition, you can add assembly components to the assembly family table. Online Help for Core applications provides detailed information about adding an assembly member to a family table. You can either suppress the table-driven assembly component in an instance of the assembly or replace it with another model. In Assembly mode, you can replace instances from the same family using the **Replace** command from the Component menu. Online Help for Core applications provides detailed information about retrieving assembly instances and their generics.

To replace components in a subassembly of an assembly instance, you must first retrieve the subassembly and edit its family table to add these components into the table and replace them in an instance of the subassembly. The subassembly itself is then replaced with its instance in the instance of the assembly.

When you want to configure (replace, suppress, or resume) many components, or components at different levels, manually editing the family tables of the assembly and the subassemblies may become tedious.

However, you can use the Assembly Instance Configurator to automate this procedure.

Access the FAMILY TABLE dialog for the assembly family table, select an instance of the assembly, and



choose **Tools > Configure Assembly Components**. A separate dialog box opens, showing the generic assembly tree at the left. You can select a cell in the column next to any component of the assembly (at any subassembly level) and type in the name of a model to replace it in the current assembly instance. You can also replace it using a model from the component family. The system displays the family of the component model and you can select a model to replace it.

When you click **OK** in the Assembly Instance Configurator, the system automatically modifies the family tables of the assembly and the affected subassemblies to provide the defined configuration of the components.

An assembly-context reference is created when features in one object use geometry from another assembly component. When an instance of the assembly is being created, assembly-context references must be updated to the instance context. Otherwise, they continue to reference the generic and therefore cannot be redefined in the assembly instance. Also, if the referenced component is replaced in the assembly instance, you must update the reference to the instance context so that the geometry of the referencing feature updates according to the changes in the source geometry.

To update the assembly references, add a reference model to the family table of the component model. (Online Help for Core applications contains an example of adding a reference model to a family table.) However, each such update involves corresponding manual editing of family tables of both models—the assembly, and the component model.

The Assembly Instance Configurator makes this updating automatic and easy. In the Ref Status column in the Model Tree, a question mark is shown next to every component model that has a reference to the generic assembly. You can update the references of an individual component model to the assembly instance or for all components at once (at all levels of the assembly tree) that still have references to the generic assembly. When you click **OK** in the Assembly Instance Configurator, the system automatically adds a reference model to the family tables of the corresponding components and updates these tables to ensure the references are correct in the assembly instance.

To update the references to all the assembly instances at once, you can use **Tools/Switch External References to Instances**. The system checks all the components included in the assembly family table. If assembly references are found in some of them, the system attempts to switch them in each model to the corresponding instance of the assembly in which these component models participate. (Online Help for Core applications contains detailed information about updating external references to assembly instances.)

The Assembly Instance Configurator can automatically update assembly references in components at any level of the assembly, but for one selected assembly instance. Update Refs can update references for all assembly instances at once, but only for components in the immediate assembly (not for subassembly components).

## Creating Reference Dimensions

You can create a reference dimension in an assembly using the same method you would use in Part mode. It can be a linear distance, a radial value, or an angular value. If you create reference dimensions on an assembly, you can then display them in assembly drawings.

Online Help documentation for Core applications provides detailed information about Part and Assembly Setup.

## Displaying Assembly Information

You can use the **Info** menu to display information. You can also use **Info** on the right mouse pop-up menu from the Model Tree to display information about the assembly.

**Note:** **Info** displays information only for objects in session. **Info** displays information about a suppressed object if it is still in session.

To display an information window providing the assembly name and the hierarchy of the names of the assembly components, you can do one of the following:

- Choose **Model** from the **Info** menu.
- Select an assembly from the Model Tree using the left mouse button, right click to display the pop-up menu, and choose **Info** and **Model Info**.

To display parent/child information, information about features, or information about parts, you can select a part from the Model Tree using the left mouse button, right click to display the pop-up menu, and choose **Info** and **Parent Child Info**, **Feat Info**, or **Model Info**.

You can use the other commands in the **Info** menu to do the following:

- Obtain mass properties of the assembly such as volume, center of gravity, and moments of inertia.
- Obtain names of files created in the current working session.
- Obtain measure information.
- Obtain information about features and parts.
- Obtain parent/child information.
- Obtain layer information concerning the entire assembly, any subassembly, or a component part.
- Regenerate the part, giving information about each feature.
- Generate a Bill of Materials.
- Show surface properties with color representation.
- Show curve properties with color representation.
- Show audit trail information about a specified model.

Online Help documentation for Core applications provides detailed information about the **Info** menu.

## To Review Assembly Instructions for a Component

Using the **Comp Info** command, you can access assembly instructions for individual components.

1. Select **Info > Component...** to display the Component Constraints dialog box.
2. Select a component. The constraints for that component appear in the Component Constraints dialog box.
3. Select a constraint from the list in the dialog box. Pro/ENGINEER highlights the corresponding model geometry in magenta and cyan.
4. Click **Apply** to display an information window with detailed assembly placement constraint information for the selected component.
5. Click **Close** to exit the Component Constraints dialog box.

## To Check Clearance

In Assembly mode, you can check clearances and interferences between parts, as well as between any two surfaces within the assembly.

**Note:** Exploded views are only cosmetic and have no effect on clearance computations. The displayed results in such cases correspond to an unexploded model.

1. Choose **Analysis > Model Analysis...** to open the **Model Analysis** dialog box.
2. Select one of the following options from the **Type** pull-down list:
  - **Pairs Clearance**—Selects subassemblies, cables, surfaces, or entities to check for clearance or interference.
  - **Global Clearance**—Finds all pairs of parts or subassemblies that are less than a specified clearance distance.
  - **Volume Interference**—Identifies all components interfering with or enclosed by a closed quilt.
  - **Global Interference**—Finds all interfering pairs of parts or subassemblies.

**Note:** Online Help documentation for Core applications provides detailed information about the **Info** menu.

## Setting Advanced Clearance Checking

You can control the calculation accuracy that Pro/ENGINEER uses for clearance checks.

The system's default method of clearance checking is to check for a local minimum at random points. You can specify a slower but more accurate method of checking for clearance and distance measures, whereby the system computes the high quality first guess based on refined triangulation. With this method, the system facets the surfaces of the model (with a quality that is proportional to the configuration file option value setting), and checks for a local minimum at each triangle point.

To perform more accurate analysis on pairs clearance, use the `clearance_triangulation` configuration file option settings:

- `none`—checks for a local minimum at random points (the default)
- `low`—corresponds to a 7 x 7 minimum grid on a surface
- `medium`—corresponds to a 14 x 14 minimum grid on a surface
- `high`—corresponds to a 21 x 21 minimum grid on a surface

**Note:** These settings affect performance and may not be acceptable for very large assemblies. Clearance interference calculations using these settings take 50 to 150 times longer than default clearance checking calculations.

## Using an Assembly Bill of Materials (BOM)

The Bill of Materials (BOM) provides a listing of all parts and parameters in the current assembly. The BOM and mass properties for the assembly are always based on the Master Representation and the components in the Master Representation.

You can customize the BOM by doing either of the following:

- Customize the text output format for a particular form of presentation and content.
- Use Pro/REPORT to create the BOM in table format in drawings.

In the assembly BOM, you must still list nongeometric assembly features that do not have representable geometry such as glue, paint, and solder (referred to as "bulk items").

### BOM and Mass Properties Behavior in Skeletons

When working with a skeleton model in an assembly, Pro/ENGINEER generates Bill of Materials (BOM) information and mass properties information that accurately reflects the design models and either the default or user-specified mass properties. However, the assembly BOM and assembly mass properties ignore skeleton models entirely when working on parts.

### BOM and Mass Properties Behavior in Master Representations

To obtain the full BOM or the mass properties of the Master Representation while working with a simplified representation, you must switch to the Master Representation. Pro/ENGINEER includes included components in mass property calculations because they are in session. It does not include excluded components unless they are in session. Mass properties only reflect what is currently on the screen. The BOM lists all components of assemblies that are in session. Unless the Master Representation is in session, the BOM is not accurate. Pro/PDM provides the full BOM without retrieval of objects. In Pro/REPORT, the BOM is available only when you retrieve an assembly as the Master Representation.

For substituted objects, Pro/ENGINEER has access to the names of both the original object and the substituted object. The mass properties of the substituted component are available because the component is in session. If they have been assigned through Interchange mode, the mass properties of the original object are available in the substituted component.

## To Specify the Model to Which Relations Apply

From the ASSEMBLY menu, choose **Relations** > **Assem Rel** to select the current assembly or a subassembly, or **Relations** > **Part Rel** to select a part.

## Entering Relations in Assembly Mode

You can add relations to parts or between parts within an assembly. Relations in Assembly mode follow the standard rules for relations except that you must first specify the model to which relations apply. This can be the main assembly, a subassembly, or a part. Once specified, all relation operations apply *only* to the specified model. For example, if the specified model is a part, the system shows only dimensions of the part in the assembly.

The notation you use to enter relations in Assembly mode differs from the notation you use to enter relations in Part mode.

In Assembly mode, for each assembly parameter, you must specify a session\_ID that refers to a component in the assembly. The relations file contains a table that specifies the session ID for each part.

For example, parameters in an assembly appear for part with session\_ID 1 as d0:1, d1:1, and d2:1; for part with session\_ID 3, they appear as d0:3, d1:3, and d2:3. If you select a part in Assembly mode, you can enter the part relations either in Part format ( $d0 = 2*d1$ ) or in Assembly format ( $d0:3 = 2*d1:3$ ).

When using relations to relate parts within assemblies, keep in mind the following:

- If the relation drives a part that is a member of the assembly, it does so only in the context of the assembly. (In Part mode, you can modify the driven value if the assembly containing the relation is not in memory.)
- You can use assembly relations to drive dimensions only on dimensions driven by a family table.
- You cannot add or edit an assembly relation that tries to drive a parameter that a part relation is already driving.
- If you add a part relation that drives the same parameter as one that already exists in an assembly relation, the new part relation drives the parameter, but the system displays an error message during the assembly regeneration.

## Showing Dimensions in Part and Assembly Modes

You can use **Show Dim** in the RELATIONS menu in both Part and Assembly modes to enter a dimension in the symbolic format. The system displays the dimension in symbolic format for the model with the given session\_ID. You can identify the session\_ID in either of two ways:

- Select a feature in the model to show dimensions; then choose **Switch Dim**.
- In the RELATIONS menu, choose **Component Id**; then select the component. The system displays the session ID of the component in the message window.

The terms session ID, runtime ID, coding symbol, and component ID are all equivalent.

## Using Assembly-Specific Configuration File Options

Many assembly-specific configuration file options are described in the appropriate topics in online Help. It is recommended that you review the available configuration file options.

Choose **Utilities** > **Preferences** to open the Preferences dialog box. You can view all the configuration file options alphabetically, and you can sort them to view the assembly-specific options. From here, you can examine and modify the `config.pro` file and generate reports on the current settings.

Online Help for Core applications provides detailed information about the Preferences dialog box.

## About Setting Component Display States

Using the **Model Setup > Component Display** command in the **View** menu, you can assign one of the following display states (visualization states) to components in an assembly:

- Wireframe
- Hidden line
- No hidden line
- Shaded
- Blanked

Components appear in the currently assigned display state. The current setting is indicated in the display status column (any column of the **Visual Modes** type) in the Model Tree window.

## To Set Component Display State

1. Choose **View > Model Setup > Component Display**. The COMP DISPLAY menu appears, with the following commands:
  - **Create**—Creates a new component display state.
  - **Set Current**—Sets the current component display state.
  - **Copy**—Copies an existing component display state.
  - **Redefine**—Edits an existing component display state.
  - **Rename**—Renames an existing component display state.
  - **Delete**—Removes an existing component display state.
  - **List**—Allows you to view existing component display states.
2. Choose **Create**, and enter a name for the component display state.
3. The system displays the Model Tree window and the EDIT DISPLAY menu.  
Choose one of these menu options from the EDIT DISPLAY menu to set the display status:
  - **Blank**—Blanks components in the current component display state.
  - **Wireframe**—Applies wireframe display state to assigned components.
  - **Hidden line**—Applies Hidden Line display state to assigned components.
  - **No Hidden**—Applies No Hidden Line display state to assigned components.
  - **Shading**—Applies Shading display state to assigned components.
  - **By Display**—Uses the display state of a subassembly in this display state. After you select a subassembly, the By Rep dialog box displays a list of possible component display states for that subassembly.
  - **Undo**—Removes the display operation on selected components. Resets the display of selected components to normal state.
  - **Info**—Displays the information window showing all assigned display states for a selected component.
  - **Definition Rules**—Sets or changes the rules that define the content and settings of the components in the display state. Online Help documentation for Advanced Pro/ASSEMBLY Extension provides detailed information.
  - **Undo Last**—Undoes the last action applied.
  - **UpdateScreen**—Displays the current state as it is defined.
  - **Display Mode**—Sets the rule for component display in the Model Tree Window. Displays the DISPLAY MODE menu.
4. Choose a command from the SELECT MDL menu to assign the specified display status state to assembly

- components.
5. Select one or more components to display in the specified display state. In the Model Tree, the current display state is indicated next to specified components in the display editing column.
  6. Choose **Done** from the EDIT DISPLAY menu, and **Done/Return** from the COMP DISPLAY menu.

## Component Display States

The current **View > Model Display** settings control the display of components that have no assigned display state as follows:

- The system displays components in front of the assigned component in wireframe and shows all of their edges in white (or the color they have), when you do the following:
  - Assign one component to display its hidden lines
  - Leave the rest unassigned
  - Set the general display style to **Wireframe**

The system displays the components behind the assigned part in wireframe, but does not display sections of edges masked by the hidden line component.
- If you set the General Display Style to **Hidden Line**, the system displays all components and the hidden line component in the same way.
- If you set the General Display Style to **No Hidden**, the system displays all components in front of the assigned component with no hidden lines, and displays the obscured edges of the hidden line component in gray and its visible edges in white. In components behind the assigned part, visible edges appear, but obscured ones do not. The edges on the other parts that the hidden line component obscures do not appear in gray.
- If you set the General Display Style to **Shading**, the system displays all unassigned components as shaded, whether they are in front of or behind the hidden line component.

The display status of the components in the assembly affects only the top-level assembly. The system stores the information about display states for an assembly in that assembly, not in the components. In an assembly, you can activate display states that you establish in a subassembly, but when you make changes to the subassembly component status at the top assembly, they affect only the display of the top assembly.

Components in a subassembly may have certain defined states before you assemble the subassembly into the assembly. Use the setting of that subassembly in the assembly by using the **By Display** command in the **Edit Display** menu. The system chooses one of the established states in the subassembly and displays it accordingly. If you change the display state of two components of the subassembly to wireframe, for example, the system displays the subassembly in its new setting, and shows the change in the top-level assembly. As a result, if you retrieve the subassembly in another session, it retains the visualization display states that you assigned to it.

When you use various display (visualization) states, keep in mind the following:

- The display state of the components in an assembly overrides the General Display Style setting. You cannot mix General Display Style settings and different states on individual components. When you select a component and assign a display state to it, the unassigned components appear in accordance with the General Display Style setting.
- You can store the established state of components with assigned display states and retrieve them by name so that you can return the model to regular display state without losing established settings.
- Only one display state can be active at one time.
- You can change display states in simplified representations and exploded assemblies as well.

## Selecting Components Using the SELECT MDL Menu

Choose a command from the SELECT MDL menu to select components:

- **Pick Mdl** (selected by default)—Selects a component from the screen or from the Model Tree. If you choose a subassembly, the system "marks" it and all of its components.
- **All**—Selects all the components in the assembly.
- **From/To**—Selects a range of models in the Model Tree.  
Selects the first and the last component in the assembly tree structure. The system "marks" these components and all the components in between. If a subassembly falls into this range, it "marks" the subassembly and all of its components.
- **By Rule**—Sets up or designates a rule for component selection using the By Rule dialog box.
- **By Rep**—Selects models active in other representations.  
Online Advanced Pro/ASSEMBLY Extension documentation provides detailed information about using the Definition Rules dialog box to select components.

## To Redefine Display Status from the Model Tree

1. From the menu bar at the top of the Model Tree window, click **Tree > Column Display...** to open the **Model Tree Columns** dialog box.
2. Select **Visual Modes** from the **Type** pull-down menu and add **Comp Display** (or any of the display states listed) to the **Displayed** column list.
3. Select **OK**. A new column labeled **Comp Display** or **VIS000#** appears in the Model Tree window.
4. Using your left mouse button, pick a display status; then select a new display status from the pull-down menu at the top of the Model Tree window. The display status of the component changes after you make your selection.

## To Redefine Component Display State

1. Choose **View > Model Setup > Component Display > Redefine**.
2. Select the name of the component display state from the Open Rep dialog box, then choose **OK**. The system adds a column to the Model Tree window labeled with the name of the display state (for example, EDIT: VIS0001).
3. Make selections from the EDIT DISPLAY menu.
4. Select the item(s) in the Model Tree window or on the graphics screen. Choose **Done** from the EDIT DISPLAY menu. The visualization state of the component changes on the screen and the column disappears from the Model Tree window.  
If the display status you need is not listed in the pull-down menu at the top of the Model Tree window, enter it in the text box.

## About Naming and Retrieving Components

Whenever the system prompts you to select a component, you can do so in two ways:

- Navigate from one component to another to select a component
- Select a specific component by name from a menu

When you assign a name to a component using the ASSEMBLY menu, the name you assign appears in the NAVIGATE ASM menu along with the original model name.

For example, if one of the components in your assembly is a bolt, the system assigns it the name `bolt.prt`. If you then proceed to add more bolts to your assembly, the component name `bolt.prt` appears only once in the NAVIGATE ASM menu, regardless of how many bolts you actually add. If you select the name `bolt.prt` from the list, the system highlights each bolt, one at a time, until you confirm a selection. However, if you assign particular component names to each of the bolts using the ASSEMBLY menu (for example, `bolt1`, `bolt2`, and so on), those names appear in the NAVIGATE ASM menu along with the original model name (in this case, `bolt.prt`).

If you know the specific component that you want, you can choose its name from the menu. If you are not certain, you can use the assembly navigation method.

## To Name a Component

You can give components in an assembly names that are easily recognizable.

1. Choose **ASSEM SETUP > Set Up > Name > Component**.
2. Select the component to name and enter the new name.

Pro/ENGINEER uses this name in both the Model Tree and in selection by menu. The system does not rename the component in the database.

The name you assign appears in the NAVIGATE ASM menu, along with the original model name. You can select a specific component by selecting its name from the menu.

## Tip: Name a Component from the Model Tree

You can name a component by entering a name in the Feat Name column of the Model Tree window.

## To Select a Component by Name from the Menu

1. Choose **Sel By Menu** from the GET SELECT menu. The NAVIGATE ASM menu appears.
2. Select the name of the component from the list (the particular name you gave it using the ASSEM SETUP menu). The system highlights the component.

The current model that contains that component is not highlighted automatically, but you can choose **HiliteCurMdl** to highlight it. If the current model and the top-level assembly are the same, the top-level assembly is not highlighted, and the menu command corresponding to it is dimmed.

## Tip: Use Query Sel to Select a Merged Part

You can use **Query Sel** to select a merged part during assembly modification.

## To Select a Component by Navigating

1. Choose **Sel By Menu** from the GET SELECT menu. The NAVIGATE ASM menu appears.
2. Select the name of the model from the list (the name you gave it when the system originally prompted you to name it during creation).
3. The system highlights every occurrence of that model, one at a time. Confirm the selection by choosing **Accept**, or choose **Next** to keep moving from one model to another until the system selects the specific one that you want.

## To Rename a Component

Using the **Rename** command from the **File** menu, you can rename a component in an assembly if both the assembly and the component are active in memory.

1. Retrieve the component that you want to rename.
2. Choose **File > Rename**.
3. From the **Model** menu, select the current name of the component.
4. Choose either **Rename in session** or **Rename on disk and in session**, and then enter the new name.



5. Regenerate the assembly.
6. Save the assembly before exiting Pro/ENGINEER.

## About Creating an Assembly

To create a subassembly or an assembly, you must first create datum features or a base component. You can then create or assemble additional components to the existing component(s) and datum features. You cannot attach components to an exploded assembly.

### Assembling Components

You can add components to an assembly in the following ways:

- Assemble a component parametrically by specifying its position relative to the base component or other components and/or datum features in the assembly.
- Assemble a component nonparametrically using the **Package** command in the COMPONENT menu. Use packaging as a temporary means to include the component in the assembly; then finalize its location with assembly instructions.
- Create a part or subassembly directly in Assembly mode.
- If you have a Pro/NOTEBOOK license, you can use layouts and specify declarations to assemble components automatically. You create these assemblies by automatically aligning datum planes and axes of different parts in accordance with the declarations previously made in Layout and Part modes. You must have a Pro/NOTEBOOK license to specify declarations; however, once a component has a declaration, it can be automatically assembled even if you do not have a Pro/NOTEBOOK license.
- With an Advanced Pro/ASSEMBLY Extension license, you can include an unplaced component. You can include a component as a member of an assembly without actually placing it in the assembly window. This technique allows you to list the component as a member of the assembly even if the component is not ready to be assembled (for example, it does not have geometry). The system lists included components in the Model Tree and BOM, but does not display them on the screen or include them in mass property calculations. To add constraints later, you can redefine the placement of the component.

You can remove a component from an assembly by deleting it or replacing it with another component. In addition, you can also redefine the placement constraints for assembled components.

To place a base component or feature, you must either create three orthogonal datum planes as the first feature, assemble an existing component (part, subassembly, or skeleton model), or create a base component.

### Using Datum Planes as the First Feature

When you create three orthogonal datum planes as the first feature in an assembly, you can assemble a component with respect to these planes, or create a part in Assembly mode as the first component. Using datum planes as the first feature has the following advantages:

- You can redefine the placement constraints of the first assembled component.
- You can pattern the first component you add, creating a flexible design.
- You can reorder subsequent components to come before the first one (if the components are not children of the first component).

### Assembling a Component Parametrically

Using the Component Placement dialog box, you can assemble components parametrically by establishing constraints that define the component's position in the assembly. The component's position changes according to changes in components or assembly features to which it is constrained.

### Creating a Base Component

If you do not create three orthogonal datum planes, the base component is the first part, subassembly, or skeleton model placed into an assembly. In many ways it is like the base feature of a part. The initial assembly units are the same as the units of the base component. When a base component is the first object in an assembly

(before any assembly features), no placement constraints are defined. The component is simply placed by default. If you replace a base component with interchangeable components, the replacing components will always be placed by default as well.

When you create the first component of an assembly, you can either create an empty component or copy from an existing component. As with an assembled base component, the initial assembly units are the same as the base component, and interchange components that replace the created base component will always be in the default orientation.

## Tip: Set Up or Use a Default Template

If you have specified a default template (you can set the configuration file option `start_model_dir` to specify its location), the system uses that template, or start model, for the part. Using a template as a start model allows you to include critical layers, datum features, and views in the model.

Click *See Also* for detailed information.

## About Placing Components

You can position a component relative to its neighbors (components or assembly features) so that its position is updated as its neighbors move or change, provided the assembly constraints are not violated. This is called parametric assembly. You can use constraints in the component placement dialog box to specify how and where the component relates to the assembly. To open this dialog box, click **ASSEMBLY > Component > Assemble** and then select the desired component from the File Open dialog box.

**Note:** If interfaces have been previously defined for the component, another dialog for selecting from existing interfaces opens before the Component Placement dialog box. If no interfaces are selected, the Component Placement dialog box opens with no constraints. Click *See Also* for more information regarding interfaces.

To assemble a component parametrically, use the Automatic constraint to select a reference on the component and assembly. For example, specify component placement by constraining a pair of surfaces to mate—the Automatic constraint type will select an appropriate constraint based on the references selected and their orientation. Using Automatic constraint type, establish references between a surface on the component and a surface on the assembly that are facing each other. The Automatic constraint becomes a mate, coincident. If the automatically determined constraint is not what you desire, you may change the constraint type.

It is also possible to select the desired constrain type first and then select relevant references. System-defined constraint types are available by using the Constraint type pull-down list in the Component Placement dialog box.

Another option would be to select the type of Offset desired first. You may specify that you want the constraint to be "Oriented" or "Offset" or "Coincident". And then select either the references you want to orient, or specify the constraint type (Automatic, Align or Mate) desired.

During component placement, you can use a combination of mouse and keyboard commands to move the active component around the screen and position it. Direct manipulation of the component can speed up the process of locating the component correctly in the assembly and establishing constraints.

The **Constraint Type** list contains the following placement constraints:

- **Automatic**
- **Mate**
- **Align**
- **Insert**
- **Coord Sys (Coordinate System)**
- **Tangent**

- **Pnt on Line (Point On Line)**
- **Pnt on Srf (Point On Surface)**
- **Edge On Srf (Edge On Surface)**

**Note:** Orient does not exist in the Constraint Type list. It is considered a subset of Align or Mate. The equivalent of an Orient constraint is an Align with Offset set to Oriented. In addition, Fix and Default constraints are not included in the Constraint Type list. Buttons are provided for both of these options in the Component Placement dialog box.

Click *See Also* for detailed information.

## Forced Coincident Align

You can force coincidence for the **Align** constraint type to force strict alignment for axes or points. Click *See Also* for detailed information.

## Pro/ENGINEER Constraint Assumptions

You can switch constraint orientation assumptions on and off. Click *See Also* for detailed information.

## Assembling Simplified Representations of Components

You can assemble simplified representations of components to a simplified representation of an assembly. After you choose the name of the new component, the system checks the component for simplified representations. If none exists, it uses the regular assembly process. If representations do exist, you can choose one from the Open Rep dialog box. You can then assemble the representation to the assembly.

**Note:** You can assemble a simplified representation of a part or assembly only to a simplified representation of an assembly. In the active representation of the top-level assembly, the system shows the component as substituted by the simplified representation that was assembled. The top-level Master Representation always contains the Master Representation of its components. In all other top-level assembly representations, the default rule determines whether to include or exclude the new component.

## Assembling Skeleton Models

You can assemble an existing skeleton model to an assembly. You can assemble one or more skeleton models to each assembly or subassembly. Assembling more than one skeleton model to an assembly or subassembly is possible only when the following configuration file option is set to **yes** (the default is **no**):

```
multiple_skeletons_allowed
```

**Note:** If a skeleton model has external references to old assemblies, these references could cause problems during the regeneration of the new assembly. It is recommended that you reuse (assemble) top-level skeleton models rather than reuse mid-level skeleton models from a different assembly.

## Assembling Bulk Items



**Note:** You can reuse existing bulk items with **Component > Assemble**.

# To Place a Component

**Note:** If you have an MDX license, the Connections area of the Component Placement dialog box is available. The Connections drop-down area is used in conjunction with the Mechanism Design Extension. For detailed information about using Connections in your assembly designs, see online Help documentation for the MDX package.

1. Click **ASSEMBLY > Component > Assemble** and then select the component to redefine, or right-click the assembly name in the Model Tree and click **Component > Assemble** from the pop-up menu. After selecting a component from the File Open dialog box, the Component Placement dialog box opens and the component appears in the assembly window. If the component being assembled has previously defined

interfaces, the Interface dialog box opens to allow you to select an interface. Click *See Also* for more information regarding interfaces.

2. Using icons in the toolbar at the top of the dialog box, specify the screen window in which the component is displayed while you position it. You can change windows at any time.
  -  **Separate Window**—Displays the component in its own window while you specify its constraints. The **Automatic** placement constraint is not available when the component is displayed only in a separate window.
  -  **Assembly Window**—Displays the component in the main assembly graphics window and updates component placement as you specify constraints.
3. The **Automatic** placement constraint is selected by default when a new component is introduced into an assembly for placement in the assembly window. Do one of the following:
  - Select a reference on the component and a reference on the assembly, in either order, to define a placement constraint. After you select a pair of valid references from the assembly and the component, the system automatically selects a constraint type appropriate to the specified references.
  - Before selecting references, change the type of constraint by selecting a type from the **Constraint Type** list. The list is started by clicking the current constraint in the **Type** box. You may also select an offset type from the Offset list. **Coincident** is the default offset, but you may also choose **Oriented** or **"0.0"** from the list. If you choose **"0.0"**, type in the value of the offset in the cell and then click **Enter**.
4. After you define a constraint, The add button (+ icon) is automatically selected, and you can repeatedly define another constraint. You can define as many constraints as you want (up to the system limit of 50 constraints). As you define constraints, each constraint is listed under the Constraints area and the current status of the component is reported in the Placement Status area as you select references.

You can select one of the constraints listed in the Constraints area at any time and change the constraint type, flip between Mate and Align with the Flip button, modify the offset value, reset an Align constraint to forced or unforced, or switch between allowing and disallowing system assumptions.

Use the Default constraint button to align the default system-created coordinate system of the component to the default system-created coordinate system of the assembly. The system places the component at the assembly origin. Use the Fix constraint button to fix the current location of the component that was moved or packaged.
5. You can select the remove (— icon) or **Preview** buttons at any time.
  - To delete a placement constraint for the component, select one of the constraints listed in the Constraints area, and then click **Remove**.
  - Click **Preview** to show the location of the component as it would be with the current placement constraints.
6. Click **OK** when the status of the component is shown as "Fully Constrained," "Partially Constrained", or "No Constraints." The system places the component with the current constraints. If the status is "Constraints Invalid," you should complete the constraint definition.

If constraints are incomplete, you can leave the component as packaged. A packaged component is one that is included in the assembly but not placed. Packaged components follow the behavior dictated by the configuration file option `package_constraints`.

If constraints are conflicting, you can restart or continue placing the component. Restarting erases all previously defined constraints for the component. You can also uncheck a constraint in the constraint table to make it inactive.

**Note:** The **Forced** check box is available only when you create an **Align** constraint type, and only when it is possible to choose to force the constraint or leave it unforced. Click **Forced** to cause strict alignment for axes or points. An alignment that is forced is listed as **Align (Forced)**.

System constraint orientation assumptions are enabled by default. You can select or clear the **Allow Assumptions** check box to enable or disable system placement assumptions.

# Placement Constraint Types

The **Automatic** placement constraint is selected by default when a new component is introduced into an assembly for placement. After you select a valid pair of references from the assembly and from the component, the system automatically selects a constraint type appropriate to the specified pair of references.

Automatic selection of a constraint type eliminates the need to select one manually from the list of constraint types and increases workflow efficiency. However, in some cases of ambiguity, you may want to change the automatic selection to another type of constraint. You can also change the constraint type before selecting any references, thereby restricting the allowed reference types.

The following placement constraints are available from the **Constraint Type** list in the Component Placement dialog box:

- **Mate**
- **Align**
- **Insert**
- **Coord Sys**
- **Tangent**
- **Pnt On Line**
- **Pnt On Srf**
- **Edge On Srf**

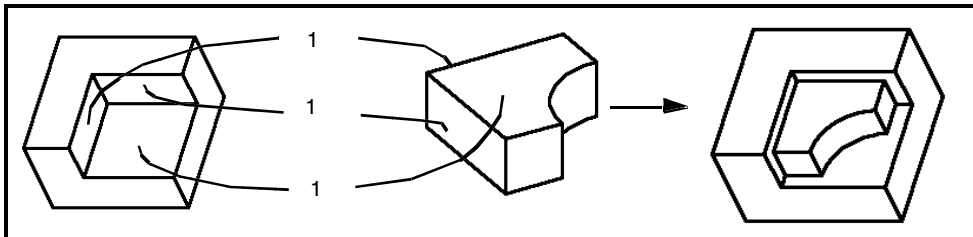
## Guidelines for Using Placement Constraints

A placement constraint specifies the relative position of a pair of references. You should follow these general principles while placing constraints:

- When you use **Mate** and **Align**, the two references must be of the same type (for example, plane-to-plane, revolved-to-revolved, point-to-point, axis-to-axis). The term *revolved surface* refers to a surface created by revolving a section, or by extruding an arc/circle. You can use only the following surfaces in a placement constraint: plane, cylinder, cone, torus, sphere.
- When you use **Mate** and **Align** and enter an offset value, the system displays the offset direction. To offset in the opposite direction, make the offset value negative.
- The system adds constraints one at a time. For example, it is not possible to use a single **Align** option to align two different holes in one part with two different holes in another part. You must define two different alignment constraints.
- Use placement constraints in combinations to completely specify both placement and orientation. For example, you can constrain one pair of surfaces to mate, another pair to insert, and a third pair to orient.

## Mate

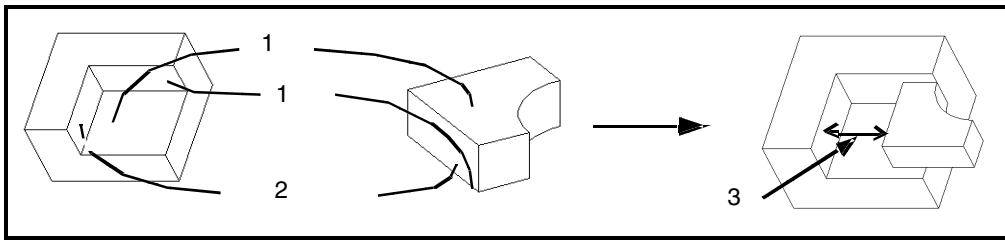
Use the **Mate** constraint to position two surfaces or datum planes with their normals pointing at each other. If datum planes are mated, their yellow sides face each other. If datum planes are mated with an offset value, an arrow appears in the assembly reference pointing in the direction where the offset is positive. If they are mated "coincident" or with an offset value of zero, the planes are coincident with the normals facing each other.



1 Mate

## Mate With an Offset Value

Use the **Mate** constraint to make two planar surfaces parallel and facing each other. The offset value determines the distance between the two surfaces.



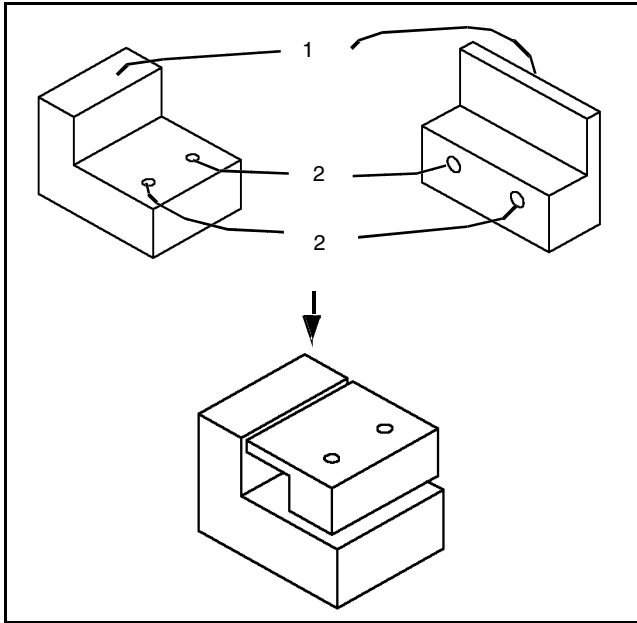
- 1 Mate
- 2 Mate offset
- 3 Offset

## Align

Use the **Align** constraint to make two planes coplanar (coincident and facing the same direction), two axes coaxial, or two points coincident. You can align revolved surfaces or edges. The yellow sides face the same direction instead of facing each other as when mated. The distance between the planes depends on whether they are aligned coincident or with an offset. The offset value is in the positive direction displayed by the arrow that appears on the assembly reference.

If two datum planes are mate-oriented, then their normals face each other so they can be offset at a value that is not fixed. They can be positioned in any location as long as their normals face each other. Align-orient works the same way, except that the normals face the same direction. When using align-mate or align-orient, you must specify additional constraints in order to rigidly position the component.

When multiple axis or point align constraints are used to assemble the component, the align constraints can have two behaviors, forced and unforced. For example, if two sets of axes are aligned, the axes of the first are made coaxial, but by default the axes of the second align are only constrained to be coplanar with themselves and with the axes of the first align. However, the **Forced** option, next to the **Constraint Type** list in the dialog box, can be used for the second align constraint. When **Forced** is selected, the axes of the second align are required to be coaxial as well. With **Forced** selected, the second align constraint is listed as **Align (Forced)** instead of just **Align** in the constraints list. If the two axes for an **Align (Forced)** are not coaxial, the constraint is considered invalid. Similar behavior occurs for point aligns. Any axis-to-axis or point-to-point align that does not have a **Forced** check box next to the constraint dropdown list is automatically required to be coaxial or coplanar. All plane and surface aligns are always required to be coplanar and never have the **Forced** option available.

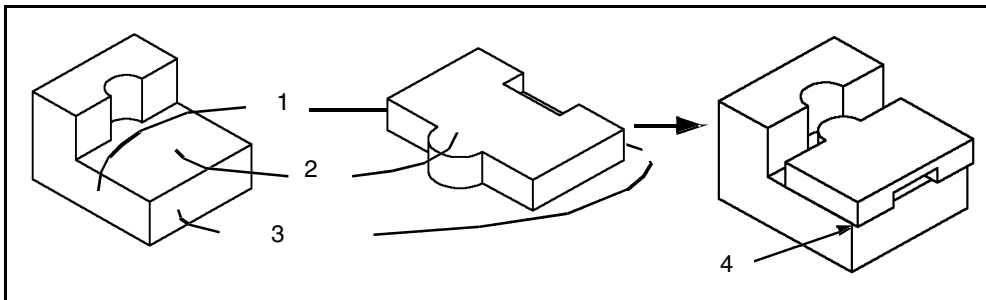


- 1 Align (plane)
- 2 Align (axis)

You can also align two datum points, vertices, or curve ends. Selections on both parts must be of the same type, that is, if you select a point on one part, you can select only a point on the other part.

### Align With an Offset Value

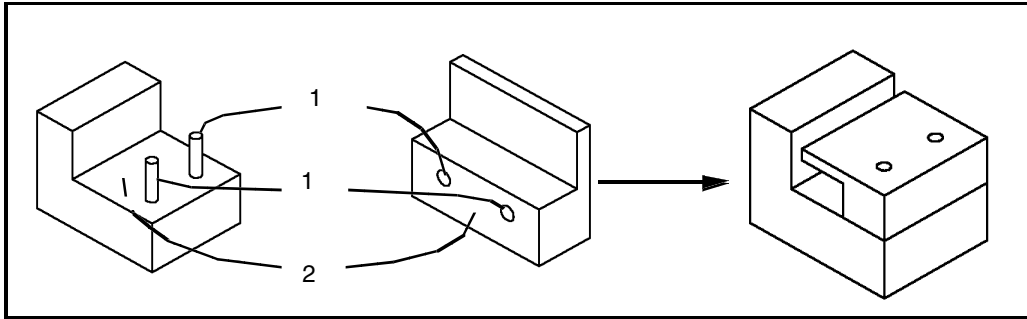
Use the **Align** constraint to align two planar surfaces at an offset: parallel and facing the same direction.



- 1 Align
- 2 Mate
- 3 Align offset
- 4 Offset

### Insert

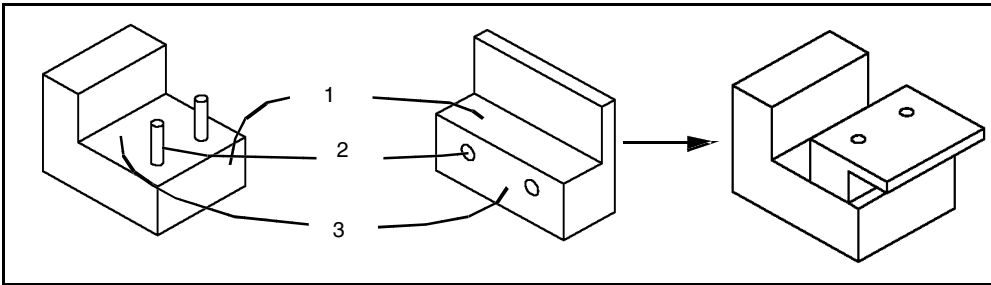
Use the **Insert** constraint to insert one revolved surface into another revolved surface, making their respective axes coaxial. This constraint is useful when axes are unavailable or inconvenient for selection.



- 1 Insert
- 2 Mate

### **Orient**

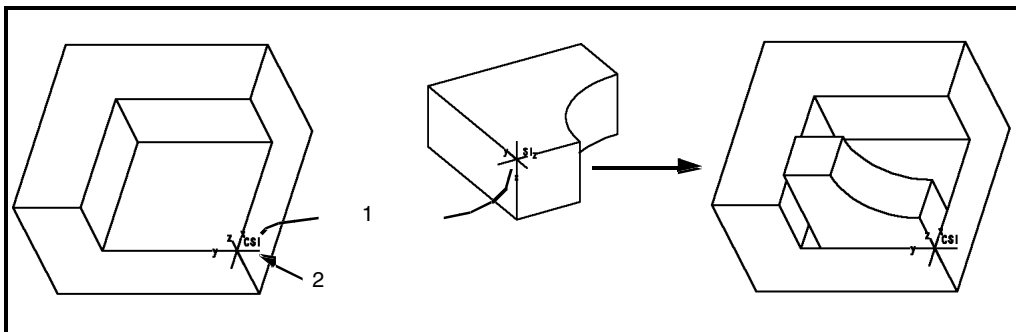
Use the **Orient** constraint to orient two planar surfaces to be parallel facing the same direction; it does not specify the offset.



- 1 Orient
- 2 Insert
- 3 Mate

### **Coord Sys**

Use the **Coord Sys** constraint to place a component in an assembly by aligning its coordinate system with a coordinate system in the assembly (you can use both assembly and part coordinate systems). You can select the coordinate systems by name from namelist menus, pick them, or create them "on the fly." The components will be assembled by aligning the corresponding axes of the selected coordinate systems.

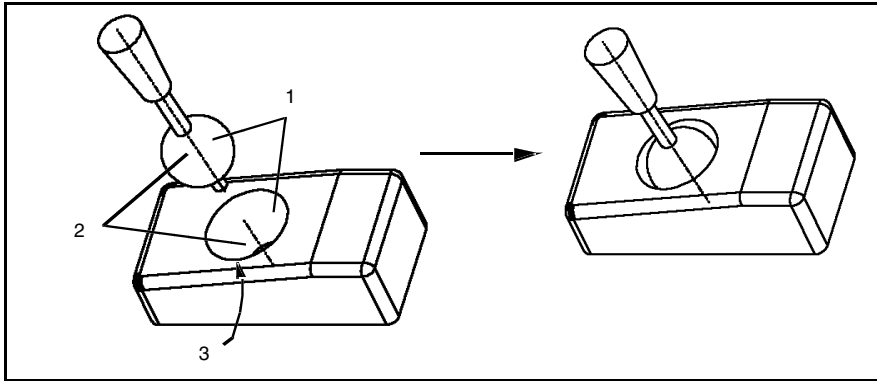


- 1 Coord sys
- 2 Coordinate system in assembly (belongs to part)

### **Tangent**

Use the **Tangent** constraint to control the contact of two surfaces at their tangency. Keep in mind that this placement constraint functions like **Mate** because it mates surfaces; it does not align them. An example of the use of this constraint is the contact surface or point between a cam and its actuator.

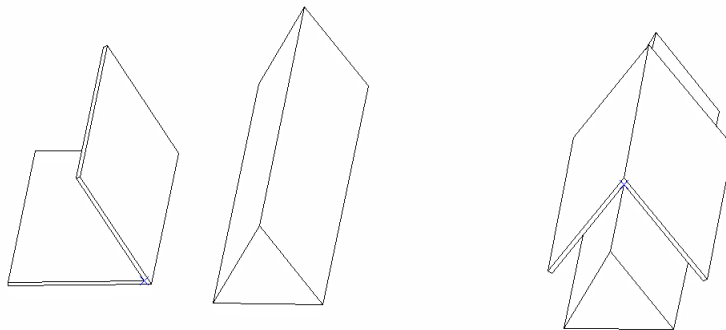




- 1 Tangent
- 2 Align
- 3 Conical surface

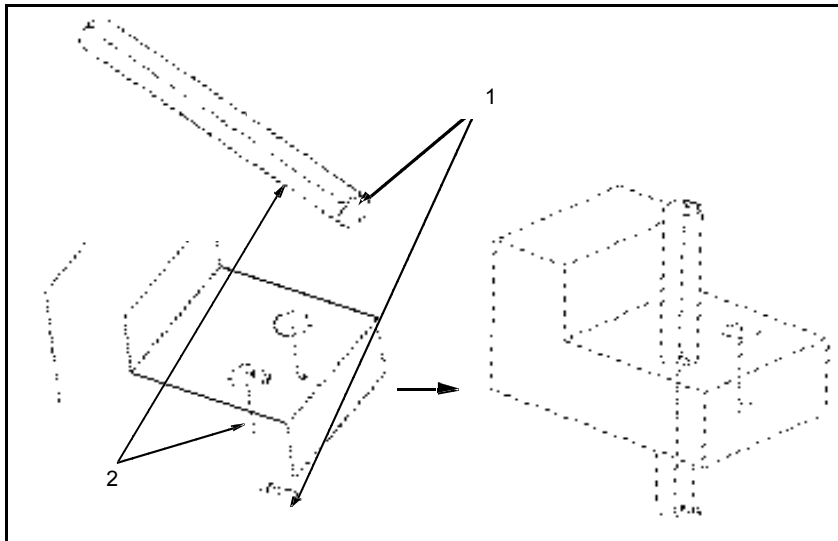
### Pnt On Line

Use the **Pnt On Line** constraint to control the contact of an edge, axis, or datum curve with a point. In the example shown in the following figure, the system aligned the point on line to an axis.



### Pnt On Srf

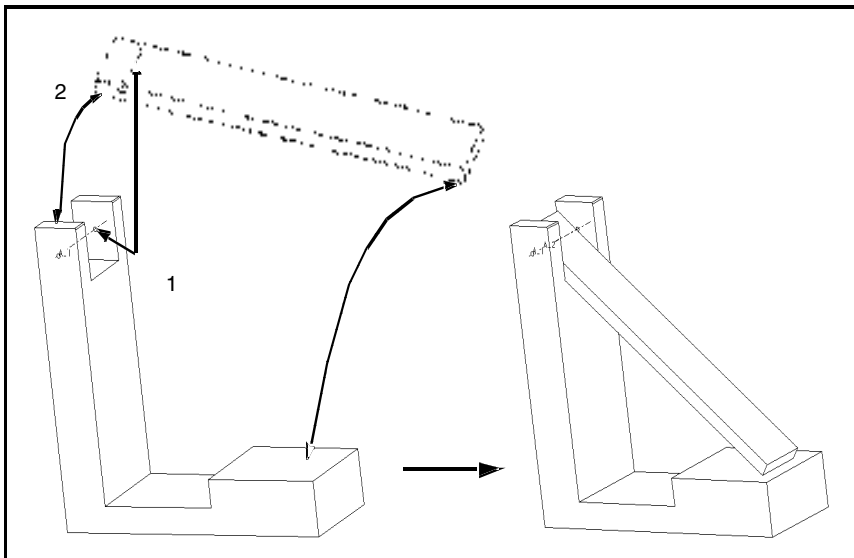
Use the **Pnt On Srf** constraint to control the contact of a surface with a point. In the example shown in the following figure, the system constrained the bottom surface of the shaft to a datum point in the hole in the block to control the depth of the shaft in the hole. You can use part or assembly datum points, surface features, or datum planes, or part solid surfaces for references.



- 1 Using Pnt On Srf, constrain this surface to be at the height of PNT0
- 2 Align both axes

### Edge On Srf

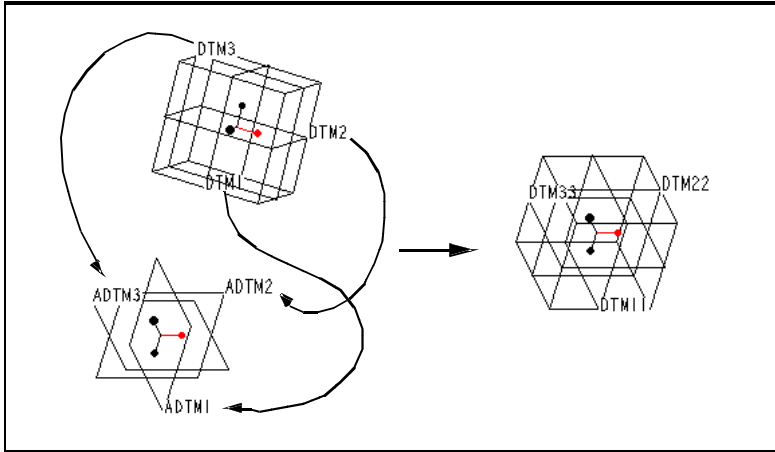
Use the constraint to control the contact of a surface with a planar edge. The following figure presents an example of a gate and pivot. The system constrained a linear edge on the pivot to a planar surface on the gate. You can use datum planes, planar part or assembly surface features, or any planar part solid surfaces.



- 1 Align using Srf, constrain edge with surface
- 2 Mate

### Default

Use the **Default** constraint to align the default system-created coordinate system of the component to the default system-created coordinate system of the assembly. The system places the component at the assembly origin.



### Fix

Use the **Fix** constraint to fix the current location of the component that was moved or packaged.

## Tip: Using Extra Constraints

You can add more constraints to a component than are necessary to place the component in Pro/ENGINEER. Even when the position of a component is completely constrained mathematically, you may want to specify additional constraints to ensure that the assembly achieves your design intent. We recommend that you limit them to 10, but the system allows you to specify up to 50 constraints.

## To Define Orient Constraints and Offset Values for Mating or Aligning

The Orient constraint option was removed from the **Constraint Type** list. To achieve the same type of constraint, use either Mate or Align and then select the **Oriented** offset option. This adds the ability to have a Mate-Oriented constraint. Achieving the Orient function by using these options allows you to explicitly indicate whether you want 2 surfaces to face each other (mate) or to face the same direction (align). If you set the constraint type to Mate and offset to Orient, the surfaces point toward each other. If you set the constraint type to Align and the offset to Oriented, the surfaces point in the same direction.

To create an Align-Oriented or Mate-Oriented constraint:

1. Select planar references so that Mate or Align appears in the Constraints Type cell.
2. Double-click the Offset cell and then click Oriented. Oriented appears in the Offset cell. The component is placed so that the references are either Align-Oriented or Mate-Oriented.

Or

1. Select **Oriented** from the Constraints Offset cell for the active constraint.
2. You can either leave the constraint type as **Automatic** or select one of the other valid constraints (Mate or Align). Select the references from the component and assembly, and the component will be constrained with either Mate-Oriented or Align-Oriented.

## To Create Datum Planes for Constraints

Placement constraints that reference planes (**Mate**, **Align**, and **Orient**) enable you to create datum planes "on

the fly" and to use them in the assembling process.

Whenever you choose one of these placement types, the system displays two options:

- **Select**—Selects a planar surface or existing datum plane; for **Align**, you can also select a revolved surface, an axis, a point, or a vertex.
- **Make Datum**—Creates a datum plane. If you choose this option after choosing a constraint type and before placing the constraint on either the assembly or the component, you must first select the window in which to create the datum. If you have already placed a constraint in one window, the system automatically creates the datum in the other window. In the component window, the system creates the datum plane as a visible feature in the component. In the assembly window, the datum is internal to the assembly.

## To Specify Direction of the Datum Plane for Mating or Aligning

If you are aligning or mating a datum plane, a yellow arrow appears on the specified datum plane by default, pointing in the direction that the yellow side currently faces.

The name of the selected reference is then shown in the **Component Reference** or the **Assembly Reference** text box, along with a yellow and a red button, each with an arrow indicating direction. You can click these buttons to flip sides from here, or you can return to the assembly window to flip sides.

## To Assemble a Component to a Datum Plane (Translate and Rotate)

You can assemble a component to a datum plane.

- To translate a component, use an "offset" datum.
- To rotate a component, use an "angle" datum.

The assembled components remain fixed relative to the datums, but you can move the datums relative to the assembly using their offset or angle parameters.

The reference datums can belong to an assembly or to a part. If they are assembly datums, their driving parameters appear in the assembly drawings; if they are part datums, they appear only in part drawings.

## To Force Alignment

Selecting the **Forced** check box, next to the **Constraint Type** list in the Component Placement dialog box, causes strict alignment for axes or points.

You can select the **Forced** check box to force coincidence for the **Align** constraint type, or clear the check box to disable it.

## Forced Coincident Align

The **Forced** check box is available only when you create an **Align** constraint type. It appears even before you select references when point-to-point or axis-to-axis types of alignment constraints already exist. If you select a plane as a reference, the check box disappears because the forced nature is assumed. Not all aligns can be left unforced; for example, the first align can never be unforced, and therefore the check box is not shown. The **Forced** check box is available whenever it is possible to choose to force the constraint or leave it unforced.

An alignment that is forced is listed as **Align (Forced)** in the Constraints area of the dialog box, in information dialog boxes, and in reports that show constraint type.

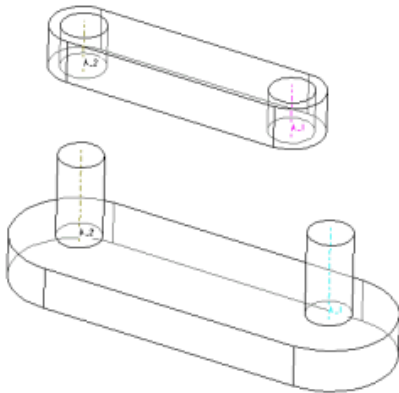
By forcing alignments, you can better capture the design intent between pairs of references that are meant to be cospatial. Should the geometry of either component change later in the design, causing the references to misalign, the placement fails, thus informing you of the constraint violation.

### Forced Disabled

When the **Forced** check box is cleared and the system locates a component by aligning two pairs of axes, the system treats each pair of axes as follows:

- First pair of axes selected—the system ensures that the axes are collinear.
- Second pair of axes selected—the system ensures only that these axes are coplanar with each other and with the axes of the first align constraint (that is, all four axes must lie in the same plane).

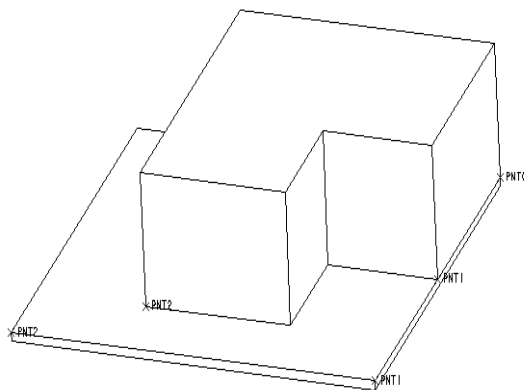
The illustration shows that the axes of the first pair are collinear (aligned), whereas the axes of the second pair are only coplanar to each other and to the first pair (unforced).



Point aligns have similar unforced behavior. If three pairs of points are aligned, the system treats each pair of points as follows:

- First pair of points selected—the system ensures that the points are cospatial.
- Second pair of points selected—the system ensures only that the points are collinear to each other and to the first pair (that is, all four points must lie on the same line).
- Third pair of points selected—the system ensures only that these points are coplanar with the first four points.

The following illustration shows three point-to-point unforced alignments.



## Forced Enabled

You can select the Forced check box to ensure that the second pair of axes is collinear. If the system cannot make these second axes collinear, the system alerts you, the alignment fails, and the component is not placed. When you use point-to-point alignment either with a secondary constraint, or even with a third constraint, the second or third set of points fails if the points are not coplanar.

## To Allow Constraint Orientation Assumptions

The **Allow Assumptions** check box, located in the Placement Status area of the Component Placement dialog box, allows you to switch system constraint orientation assumptions on and off. The **Allow Assumptions** check box is available whenever assumptions are made or could be made; when a component is fully constrained, the check box disappears. The setting of **Allow Assumptions** is component specific, and the setting is saved with the component.

## Constraint Orientation Assumptions

When **Allow Assumptions** is selected (the default) during component assembly, Pro/ENGINEER makes constraint orientation assumptions to help you assemble components efficiently. For example, to assemble a bolt to a hole in a plate, only one align and one planar constraint are required to fully constrain the components. After defining an **Align** constraint between the axes of the hole and the bolt and, for example, a **Mate** constraint between the bottom face of the bolt and the top face of the plate, the system assumes a third constraint. This constraint controls rotation about the axes, thereby fully constraining the component.

When the **Allow Assumptions** check box is cleared, the system requires the explicit definition of the third constraint before considering the component fully constrained. You can leave the bolt packaged, or you can create another constraint that explicitly constrains the bolt's rotational degree of freedom.

With the system assumptions disabled, you can package drag a component out of a previously assumed position, and have it remain in the new position. The component automatically snaps back to the assumed position if you select the **Allow Assumptions** check box again.

**Note:** Prior to Release 2000i, assemblies were created using assumed constraints.

## To Rotate a Placed Component

When system assumptions are enabled, the system fully constrains a component if it is completely placed and has only a rotational freedom of motion remaining.

You can rotate the component about this degree of freedom; however, it will snap back to the default orientation.

## To Move a Component to a New Rotation

You can rotate a placed component with an assumed constraint, and the component stays in its current rotation; that is, the component remains where you put it.

Clear the **Allow Assumptions** check box in the Placement layout of the Component Placement dialog box, and move the component using one of the following methods:

- With the placement layout of the dialog box displayed, manipulate the component using the mouse and the keyboard commands **Ctrl + Alt**.
- Click the icon to display the motion layout of the dialog box, and move the component around, and click **OK**.

The component also remains where you put it when you use the **Package** command to move it.

## To Translate and Rotate a Component About an Axis

Once you have parametrically placed a component in the assembly, you can move it with respect to a coordinate system.

1. Choose **Modify > Mod Assem > Move**. Select or create an assembly coordinate system.
2. Select the component(s) to move; then choose **Done Sel**.
3. Choose **Translate** or **Rotate** from the MOVE menu. Select the axis about which to translate/rotate the component.
4. Enter the distance of translation along the selected direction, or the angle of rotation about the selected axis.

## About Freeform Mouse-Driven Component Manipulation

Whenever the Component Placement dialog box is available for placing a component or redefining placement constraints, a spin center for the active component is always visible, and you can manipulate the active component using a combination of mouse and keyboard commands (**Ctrl + Alt**, pressed at the same time). Manipulating a component is easier than moving to a separate layout in the Component Placement dialog box to package-move a component around the screen. You can switch between full view navigation and component manipulation easily with the **Alt** key.

You can perform translation and rotation adjustments while you establish constraints. The component motion respects any constraints as they are established.

The spin icon appears during the entire component placement operation and defaults to the bounding box center. You can modify this location using the Preferences dialog box, accessed from the Component Placement dialog box.

Because of the orthographic projection used by the system, motion in the z-axis, or screen normal, is noticeable only if objects intersect. Thus, while a component is moving in the z-axis, the camera angle is adjusted to provide a noticeable effect.

In Rotate mode, when **View Plane** is the selected motion reference, the spin center does not use the screen normal spinning, but rather the two orthogonal axes (within the screen plane).

The system automatically sets drag origin. The system uses the screen selection point as the origin of rotation; therefore, you do not have to use the **Drag Center** option in the Preferences dialog box to redefine the drag origin continually.

The default spin center, or drag point, is the model center, not the default coordinate system. You can use the Preferences dialog box to change the drag origin.

## To Manipulate a Component

During component placement, you can manipulate a component's location in an assembly while establishing constraints.

1. Select a component to be placed using one of the following methods:
  - Choose **ASSEMBLY > Assemble** to place a new component.
  - Choose **ASSEMBLY > Component > Redefine** to place a previously assembled component.
2. Package moving the component is similar to standard view navigation. Manipulation is accomplished using a

combination of mouse commands and the keyboard commands **Ctrl + Alt** (pressed at the same time).

Use the mouse buttons to perform the following actions:

- **Left**?z-axis motion (moves component normal to the screen)
- **Middle**?Rotate
- **Right**?Pan (moves component within the plane of the screen)

**Note:** When there are no constraints, a z-axis motion would not be noticeable because of the orthographic projection of the graphics screen; therefore, the camera angle is adjusted slightly to make it easier for you to see the motion.

After you define some constraints, mouse/keyboard manipulation is restricted as well.

## To Set Preferences for Component Manipulation

1. Select **Preferences** from the Move tab in the Component Placement dialog box. The Preferences dialog box opens.
2. In the Drag options area of the dialog box, select any or all of the following options:
  - **Dynamic Drag** (selected by default)—Drag the component while maintaining existing constraints
  - **Modify Offsets**—Modify offset dimensions while dragging.
  - **Add Offsets**—Add offset dimensions to **Mate** or **Align** constraints initially created without offsets.
3. In the Drag Center area of the dialog box, select one of the following options to control drag origin:
  - **Model Center** (selected by default)—The spin center, or drag point, is the model center.
  - **Default Csys**—The spin center is the default coordinate system.Use the selection button to select a coordinate system to set as the default.

## To Set Snap Preferences

1. Select **Preferences** from the Move tab in the Component Placement dialog box. The Preferences dialog box opens.
2. Click **Snap Options** to display a drop-down area of the dialog box. Select any or all of the following:
  - Select **Activate Snapping** to enable snap during free form movement.
  - In the **Distance** text box, change the default value 0.100000 (units are a fraction of the screen size).
  - In the **Angle** text box, change the default value 30.000000 (units are degrees).

## Snap Assembly by Proximity

During component placement, Pro/ENGINEER's drag, snap, and drop capability allows fast constraint generation, resulting in extremely efficient assembly. Snap functionality provides intelligent feedback about the placement of a component in an assembly, allowing more efficient workflow and better managed references. Finding the correct set of constraints to assemble a component can sometimes be tricky, even for an experienced user. Snap functionality saves you from having to make a large number of mouse clicks during standard assembly, especially when dealing with standard parts such as fasteners and bolts.

After you select one reference, you can drag the component to its approximate location in the assembly. The system establishes a best-guess constraint type between the selected reference and any potential matching reference within the allowed proximal distance and angle ranges. If the system finds a matching reference, the system provides a suggested snapped placement. The system highlights both references when a potential match is found.

The component snaps to respect the suggested pair of references as long as the mouse remains within a "play region" of screen motion, as in Sketcher mode. You can accept the snapped placement by simply releasing the component or refuse it by dragging the component beyond the proximity region.

Snap functionality, used together with automatic selection of constraints (the **Automatic** placement constraint),



allows you to establish constraints easily, using the mouse, without having to return to the Component Placement dialog box. For example, when you assemble a bolt to a plate with many holes, snap and autoselection allow you to align and then mate or align the bolt to the plate with the mouse. You can select the axis of the bolt and slide it close to the axis of one of the holes on the plate; and then release the mouse. The bolt snaps to the axis of the hole, and the system automatically establishes the constraint. Now that the bolt is aligned, it is in position for the next constraint. You can select a surface on the bolt and slide the bolt along the axis of the first constraint until it is close to a constraint to which it can mate or align. When you release the mouse, the new constraint is established automatically.

The snap reference table shows the criteria that the system uses in establishing a best-guess constraint type to match the selected reference and any potential matching reference within the tolerance allowances for the following constraint pairs:

User-Selected Reference Type	System-Matched Reference Type
Plane/Face	Plane or face for Mate or Align
Axis	Axis (possibly linear edges) for Align
Coordinate system	Coordinate system for Align
Revolved surface	Revolved surface for Insert
Cylinder	Revolved surface as above; cylinder, sphere, plane for Tangent
Cone	Revolved surface as above; cone for Mate
Other edges/curves	No match
Other surfaces	No match

**Note:** Snap is not available for the following types of constraints:

- **Tangent**—for surfaces other than Plane/Cylinder/Sphere
- **Point on surface**—for nonplanar surfaces

## Tip: Enable Snap from the Preferences Dialog Box

Snap functionality is available during component placement when it is enabled. To switch between allowing and disallowing snap functionality, select or clear **Activate Snapping** in the Preferences dialog box, accessed from a toolbar button on the Component Placement dialog box.

## To Set Proximity Tolerance Allowances

To adjust the proximity sensitivity used for snap assembly, set the following configuration file options:

- Set `comp_snap_dist_tolerance` [0 to 1]. The snap tolerance is based on a fraction of the camera view. This maintains a consistent snap distance based on camera zoom.
- Set `comp_snap_angle_tolerance` to a value in the range of 0 to 90 to set the allowed angle between the normals of two planes, or between two axes. This angle will be from 0 to 90 degrees, allowing for an almost fully orthogonal snap.

## About Assembling a Component to a Pattern

You can pattern a component in the following ways:

- **Reference Pattern**—Assemble it to the leader of an existing component or feature pattern; then pattern it

using **Ref Pattern**. A pattern *must* exist in order to use this option.

- **Dimension-Driven Pattern (Nontable)**—Create the nontable pattern as a pattern by itself using **Dim Pattern** with placement dimensions from the constraints applied. You can create a nontable pattern using the same model for each member of the pattern.
- **Dimension-Driven Pattern (Table)**—Create the table pattern as a pattern by itself using **Dim Pattern** with placement dimensions from the constraints applied.
  - **Same model**—You can create a table pattern using the same model for each member of the pattern.
  - **Different model**—You can create a table pattern using different models for each member of the pattern.

If a component was created in the context of an assembly with the **Create First Feature** option, you cannot pattern it.

## To Replace a Pattern Member by Family Table in a Pattern

When you replace a pattern member by family table in a pattern, the entire pattern, including the pattern leader, is replaced even though you selected only a single pattern member. When you select a pattern or group member to add a new item into the family table, the system warns you that the entire pattern will be replaced, not just the pattern member, and prompts you to confirm the replacement. When you confirm, the pattern or group leader is added into the family table.

## To Assemble Components to a Reference Pattern

1. Retrieve a component for assembly by choosing **ASSEMBLY > Component > Assemble**.
2. Place the component using the Component Placement dialog box. The references you select from the assembly must be from a previously patterned assembly component or assembly feature.
3. Choose **COMPONENT > Pattern**; then pick the component to pattern. Or, select an assembly or component in the Model Tree, and choose **Pattern** from the pop-up menu to display the PRO PAT TYPE menu.
4. Choose **PRO PAT TYPE > Ref Pattern**. This command does *not* appear unless the component is assembled relative to an existing pattern.
5. Choose **Done**.

The component assembles to every member of the pattern. Note that placement constraints that do not belong to the pattern are common to all components. If you modify one component, the system modifies every component as in a feature pattern.

## Assembling Components to a Reference Pattern

After assembling a component, you can pattern it using the **Ref Pattern** option, as long as it was assembled with constraints that reference some previously patterned item. For example, to place bolts into a pattern of holes, you have to specify the constraints for assembly only once, between the bolt and the first, or leader, hole.

When assembling components to a reference pattern, keep in mind the following rules:

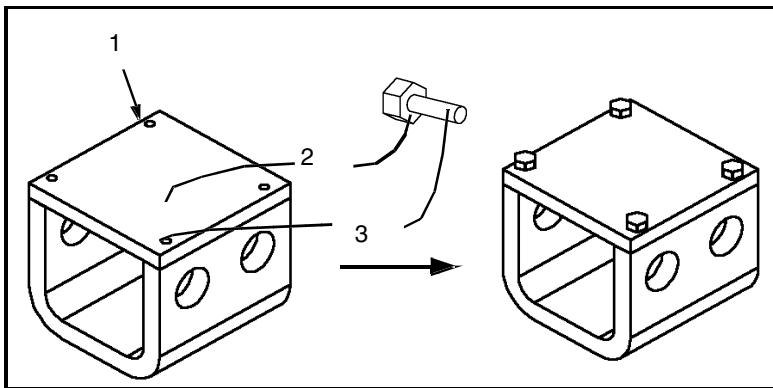
- You can select references from two patterns in a set of placement constraints, but the system uses only the first pattern for subsequent components.
- If you assemble a component to an assembly feature that is the leader of the pattern, you cannot reference

this pattern.

- You can use assembly components placed as a reference pattern to assemble another pattern of components. For example, if you have bolts assembled in a pattern of holes using the **Ref Pattern** option, you can use **Ref Pattern** again to assemble a pattern of nuts directly to the bolts.

## Example: Assembling a Component to a Pattern

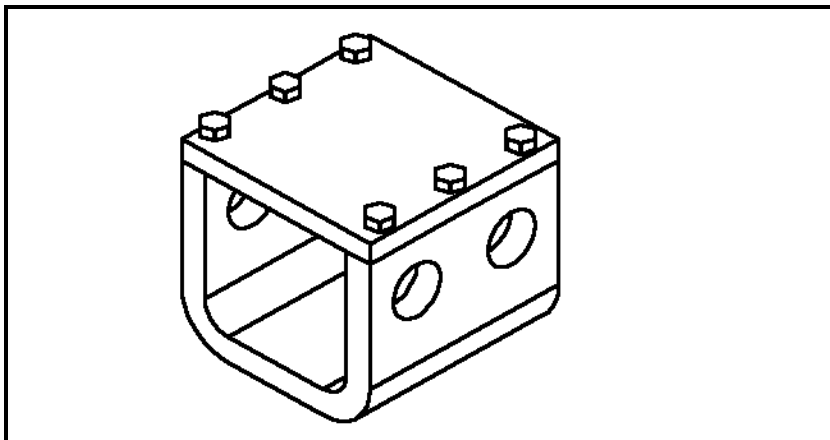
The figure below shows assembling a component to a pattern:



- 1 Pattern of four holes
- 2 Mate
- 3 Insert

The number of instances (holes) of the pattern determines the number of components (bolts) to be assembled. Therefore, if you modify the number of instances (holes) in the pattern, the number of required components (bolts) is updated.

The figure below shows modifying a pattern. The number of instances has been changed to 6. The number of bolts automatically follows the number of holes.



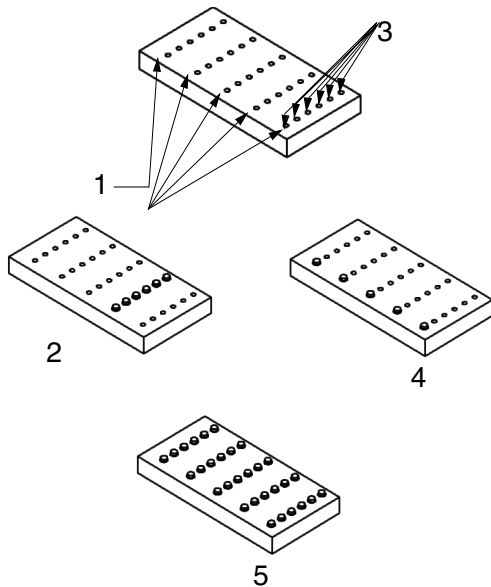
## To Assemble Components to a Group Pattern

For a pattern assembly, you can use part features that were made into a local group before being group patterned. If the assembling references contain more than one pattern, the system uses the pattern of the first reference. If the first reference contains both a feature pattern and a group pattern, the ASSEM PAT menu appears with the following options:

- **Feat Pat**—Assembles the component only to the selected feature pattern.
- **Group Pat**—Assembles the component only to the group pattern.
- **Both**—Assembles the component to both the feature and group patterns.

## Example: Assembling to Group Patterns

The figure below shows assembling to group patterns:



- 1 Group pattern
- 2 Assembled to feature pattern
- 3 Feature pattern made into local group
- 4 Assembled to group pattern
- 5 Assembled to both patterns

## To Create a Dimension Pattern Using the Same Model

1. Retrieve a component for assembly by choosing **ASSEMBLY > Component > Assemble**.
2. Place the component using the Component Placement dialog box options with at least one offset type constraint. You must specify the offset type constraints for surfaces that you want to use for creating the pattern. You must also assemble the component to a component that is already patterned; for example, assemble a bolt, referencing for placement constraints to a hole that is a member of a hole pattern.

3. Choose COMPONENT > **Pattern**.
4. Select the component to pattern. The PRO PAT TYPE menu appears when the component is assembled to a component that is already patterned.
5. Choose PRO PAT TYPE > **Dim Pattern** > **Done**. The PAT DIM INCR menu appears.
6. Choose one of the following to establish the pattern directions and increments as you would for a feature pattern:
  - Choose **Table** to create a table-driven pattern.
  - Choose **Value** to create a nontable-driven pattern.

## Dimension Patterns

You can create a table or nontable pattern using the same model for all members of the pattern.

You can create a dimension-driven pattern to specify multiple occurrences of a component in the assembly. The dimension-driven pattern uses assembly constraint dimensions to create this type of pattern, so you must use constraint types such as **Mate Offset** or **Align Offset**. You can also use **Copy** to create a pattern type assembly. The rules for creating a dimension-driven pattern for an assembly follow those for feature patterning in Part mode.

You can use the Model Tree to create a dimension-driven pattern. Choose **Pattern** from the pop-up menu in the Model Tree window to display the PAT DIM INCR menu or the PRO PAT TYPE menu.

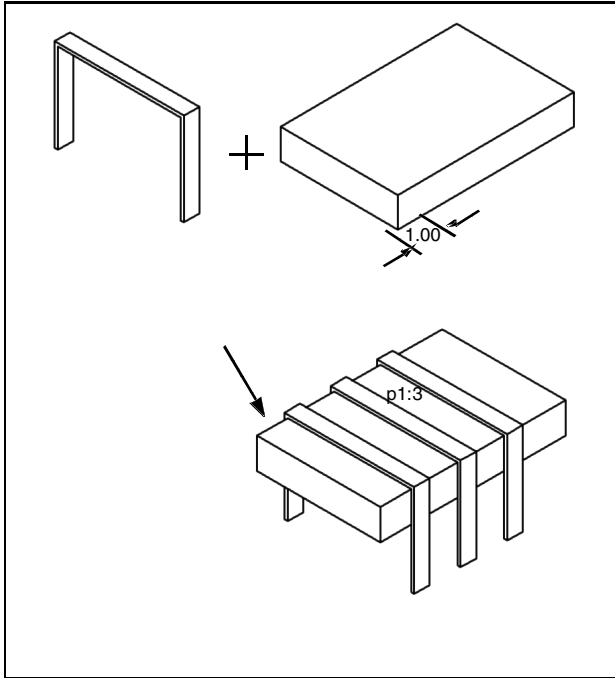
## Tip: Redefine and Recreate a Dimension-Patterned Component

To redefine a component that is patterned using **Dim Pattern**, you must recreate the pattern after redefining the component placement.

## Example: Assembling Components Using Dim Pattern

When assembling with the **Dim Pattern** option, use the dimension entered for the **Align Offset** constraint (1.00), as the driving dimension.

The resulting part was assembled only once. To change the number of components in the pattern, modify the p1 : 3 parameter.



## To Create a Dimension Table Pattern Using Different Models

In Assembly mode, you can create a pattern with different models throughout the pattern. This kind of pattern can be created only when each pattern member model and the model of the pattern leader all belong to the same family table of models.

When you have patterned a component in an assembly using a pattern table, you can replace individual occurrences of the patterned component with members of its family table by editing the pattern table file and specifying a family table instance name in the Model column. The Model column appears in the pattern table of a newly created component pattern—patterns created in Release 2000i or later—when the model of the pattern leader belongs to a family of models, that is, when the model has family table instances or is a family table instance itself. You can replace any or all individual occurrences in the pattern table with instances.

Be sure to use a common reference for the initial placement of the component. The family table member must have the component reference used by the driving pattern component. If any of the component's instances do not contain all the placement references used by the pattern leader, these instances cannot be placed in the pattern, and the pattern fails. You can then redefine the placement references of the pattern leader so that it uses references common to all instances of the pattern, or make the necessary references available by editing the family table.

1. Retrieve a component for assembly by choosing **ASSEMBLY > Component > Assemble**.
2. Use the Component Placement dialog box options to place the component, with at least one offset type constraint. You must specify the offset type constraints for surfaces that you want to use for creating the pattern.
3. Choose **COMPONENT > Pattern**.
4. Select the component to pattern. The **PRO PAT TYPE** menu appears.
5. Choose **PRO PAT TYPE > Dim Pattern**.
6. Choose **PAT DIM INCR > Table**.
7. Select a dimension to pattern.
8. Choose **Done** from the **EXIT** menu. The **PATT TABLE** menu appears.
9. Choose **Add**, and enter a name for the table.

- The Pro/Table appears. If the component has family table members, the Model column also appears.
10. The selected dimensions and the Model column appear with their current values, and you can set different models as pattern members in this column. Add pattern members and fill in the new dimensional values and model names.

## Assembly Intersections in Table Patterns Using Different Models

For assembly-level visible assembly intersection features, all intersected pattern members are shown with correct assembly cuts, and each pattern member has its own intersection, which is visible only in the top assembly scope. For part-level intersection, assembly cuts appear in each occurrence (because it is the same model), just as for regular assemblies with multiple occurrence of the same models.

When an assembly feature encounters a new instance and you manually identify it as an instance to be intersected, to avoid conflicts between pattern table and assembly feature definition, the system behaves as follows:

- When the intersected component is a pattern member, the new instance name is displayed in the pattern table.
- When the intersected model is the pattern leader, all \*'s in the Model column of the current pattern table are replaced by the original pattern leader name.

## About Packaging Partially Constrained or Unplaced Components

When you are adding a component to an assembly, you may not know where that member fits best, or you might not want to locate it relative to other geometry. You can leave such a component either partially constrained, or unconstrained. This component would be a *packaged* component. Although it is a part of the assembly, it is not assembled parametrically. Use packaging as a temporary means to locate the component; finalize the location of the component with assembly constraints. One way to package a component is to use the PACKAGE menu. Another way to package a component is to assemble the component, then close the Component Placement dialog box before the component is fully constrained.

Another alternative is to place a component by referencing a packaged parent. The system allows you to create placement references to fully constrained components, thus allowing you to begin establishing an assembly design early, before all placements constraints are known for all parent components. A unique Child of Packaged icon, similar to the Packaged Component icon, is used in the Model Tree to mark a component that is placed with a reference to a packaged parent, and the component is shown as *Child of Packaged* in the Model Tree status line and Status column. The icon is used only for first-level children of a packaged component.

You can use configuration file options to activate or deactivate the capability to reference packaged components.

As a design grows, the placement of children of packaged components may not remain as you intended because of the extra degrees of freedom. To help reduce these effects, you can use the **Fix** constraint to *fix*, or fully constrain, a packaged component in its current location, in relation to its parent assembly.

**Note:** Load a subassembly independently if you want a more immediate coordinate system, closer to the active packaged components.

You can reposition a packaged component using either the **Move** command on the PACKAGE menu, or the **Move** layout available from the Component Placement dialog box.

Using package move functionality, you can move a fully constrained component by specifying preferences, and adding and removing offsets. You can change the placement of a component, regardless of how it was

assembled, without having to redefine it.

When you package a component, Pro/ENGINEER attaches it to the mouse cursor. Move the mouse to position the packaged component and click the left mouse button to drop the part at the current position.

Pro/ENGINEER remembers where packaged components are located. However, positioning is absolute, not relative to other components.

When you select a component to drag, a model center icon is displayed at the center of the component (that is, at the center of the component's bounding box) and moves with the component as it moves, accurately representing the current position of the component. This icon is exactly half the size of the spin center icon. When the spin center icon display is turned off, the model center icon also does not display. The model center icon is independent of the drag center used in packaging.

## To Package a New Component in an Assembly

Using the **Add** command in the PACKAGE menu, you can place components into an assembly nonparametrically (that is, without constraining them relative to neighboring parts).

**Note:** The first component of an assembly cannot be a packaged component. However, you can package additional occurrences of the first component.

1. Choose ASSEMBLY > **Component** > **Package** > **Add**. The GET MODEL menu displays the following options:
  - **Open**—Opens the File Open dialog box to select a component..
  - **Sel On Model**—Allows you to select any component on the screen and adds a new occurrence of it to the assembly.
  - **Sel Last**—Adds the last component assembled or packaged.
2. Choose an option from this menu.
3. Select a component. The Move dialog box opens.  
The system automatically places the component in the default position. The component then follows the cursor in a *dynamic drag* mode, ready for you to place it.
4. Click the left mouse button to drop the component in the current position.
5. Use the Move dialog box to adjust the position of the packaged component.

## Configuration File Options for Package Moved Components

The following configuration file options control the behavior of *package moved* components:

- `comp_assemble_start`—Controls where the component is initially shown. The values are `package` (the default) and `constrain_in_window`.
- `package_constraints`—Controls the behavior of partially constrained components. You cannot build children from them. The values are `update`, `freeze`, and `disallow`.  
If you set it to `update`, the component continues to follow the assembly constraints you have specified. If you set it to `freeze`, the component behaves the same as a packaged component; that is, it does not follow the constraints that you have specified. If you set it to `disallow`, the component must be fully constrained before you can leave the package interface.
- `comp_rollback_on_redef`—Controls whether the system rolls back the assembly when you redefine a component. The values are `yes` (the default) and `no`.
- `allow_package_children`—Controls the capability to reference packaged components. If you set it to `feature`, you can make only feature references to packaged components; the `all` setting (the default) allows both feature and placement references to packaged components; the `none` setting disallows both feature and placement referencing to packaged components.

You can also use the Ref Control dialog box to switch between allowing and disallowing referencing



packaged components.

- **package\_ref\_alert**—Controls whether the system displays a confirmation prompt whenever you select a placement reference to a packaged component.

Referencing packaged components can result in "loose," or unfixed placements that can cause unexpected placement behavior after minor geometric modifications. Therefore, whenever you select a placement reference to a packaged component, the system displays an alert and requests confirmation that you want to use the reference. The values are **yes** (the default) and **no**.

- **spin\_with\_part\_entities**—Controls whether datum planes, axes, and coordinate systems move with the components when you are placing or package moving components using the mouse. The values are **yes** (the default) and **no**.

While dragging components, it is useful to obtain visible feedback about their position based on datum references, especially for components that contain only datum features and no solid geometry.

## To Move a Packaged Component in an Assembly

1. Choose **PACKAGE > Move > Preferences**. The Preferences dialog box appears.
2. Select the drag options for the move and modify the drag center, then close the Preferences dialog box.
3. Set the Motion Increments for the move.
4. Set the Transformation and Direction for the move as follows:
  - Using **Translate** or **Rotate**, and an appropriate Direction, select a component. The component moves with or rotates around the cursor dynamically and you can "drop" it into place using the left mouse button, return it to its original location using the middle mouse button, or switch between **Translate** and **Rotate** mode using the right mouse button.
  - Using **Adjust** and a planar direction, select a surface on the packaged component to align to the reference plane in the assembly.
  - Using **Adjust** and **Entity/Edge**, align the packaged component with the assembly by selecting a reference entity (an existing curve, edge, or axis) on the assembly and another entity (edge or axis) on the packaged component.
  - Using **Adjust** and **2 Points**, align the packaged component with the assembly by selecting two points on the assembly to specify the motion reference direction, then selecting an edge or axis on the packaged component.
  - Using **Adjust** and **C-sys**, align the packaged component with the assembly by selecting a coordinate system axis on the assembly, then selecting an edge or axis on the packaged component.

### Notes:

The **Adjust** option is not a rigid assembly constraint. It simply moves the component nonparametrically. It is analogous to a **Mate** or **Align**.

The **Translate**, **Rotate**, and **Adjust** options are modal. Once you select a reference in an assembly, you can switch between these options until you select the reference again. The reference remains active until you change it.

5. Continue repositioning packaged components, placing new members, or changing views.

## Moving Packaged Members

You can use the **Move** dialog box to translate or rotate assembly members that you positioned using the **Add** option, as well as components that were left with incomplete constraints.

Keep in mind the following:

- Once you add a component to an assembly, the system does not remove it when you choose **Quit**. It restores it to its original default placement. Pro/ENGINEER will remove the added component from the assembly if you quit the initial package move dialog box.
- As you move a component, the system records each movement until it completes the placement. You can use the **Undo** command until the component reaches its initial position. You can also use **Redo** in the same

way.

- When using **Adjust** and **View Plane** to move a component, the system does not move the component anywhere because the actual view plane is located very far away from the assembly. Instead, it reorients the component so that the desired surface is perpendicular to the view direction.

The Move dialog box is similar to the Move layout available on the Component Placement dialog box.

There are several areas on the Move dialog box, as described below:

- **Motion Type area**—Allows you to select one of the following radio buttons to determine the kind of motion:
  - **Translate**—Moves the packaged component by dragging it parallel to an edge, axis, plane, or the viewing screen; perpendicular to a plane; or until a face or axis on that component becomes coincident with another.
  - **Rotate**—Rotates the packaged component about an edge, axis, or point on the viewing screen; or until a face or axis on that component becomes aligned with another.
  - **Adjust**—Aligns the packaged component to a reference entity on the assembly.
- **Motion Reference area**—Allows you to select the reference for the direction from the following options:
  - **View Plane**—Uses the viewing plane as the reference plane (repositions the component in a parallel plane).
  - **Sel Plane**—Selects a plane other than the viewing plane as the reference plane (repositions the component in a plane that is parallel to it).
  - **Entity/Edge**—Selects an axis, straight edge, or datum curve (repositions the component in a line parallel to it).
  - **Plane Normal**—Selects a plane as the reference plane and repositions the component in a line that is normal to it.
  - **2 Points**—Picks two points or vertices (repositions the component in a line that connects them).
  - **C-sys**—Selects a coordinate system axis (repositions the component in the direction of it).
- **Motion Increments area**—Allows you to set the degree of the following increments (to drag the component without apparent incrementing, choose **Smooth**.):
  - **Translation**—Specifies the motion increments for translational dragging. Use the menu to select a value, or enter a value.
  - **Rotation**—Specifies the motion increments for rotational dragging. Use the menu to select a value, or enter a value.
- **Position area**—Allows you to enter the relative distance from the start point to the new component origin.
- **Undo button**—Allows you to **Undo** the last motion.
- **Redo button**—Allows you to **Redo** the last motion.
- **Preferences button**—Sets up preferences for dragging packaged components. Displays the Preferences dialog box with the following options:
- **Dynamic Drag**—Dynamically updates the component to constraints while dragging. This option is the default.
  - **Modify Offsets**—Modifies the offset dimensions while dragging the component.
  - **Add Offsets**—Adds offset dimensions to **Mate** and **Align** constraints initially created without offsets.
  - **Drag Center**—Selects a new point on the component to be the drag origin

## To Fix the Location of a Packaged Component

1. Choose **ASSEMBLY > Component > Package**. The **PACKAGE** menu appears.
2. Select the packaged component to place.
3. Choose **Fix Location**.

The system fully constrains the packaged component in its current location.

## To Finalize Packaged Components

Packaged members are not located *parametrically* in the assembly. That is, changes made to neighboring parts do not drive their location. This makes packaging useful when you are laying out your assembly and experimenting with different configurations. However, once you know where components go, you should finalize their location using a command from the ASSEMBLY menu; select either **Component**, then **Package**, then **Finalize**, or **Component** then **Redefine**.

When you finalize a packaged member, you can use placement constraints (such as **Mate**, **Align**, **Insert**, **Orient**) to reposition it in the assembly. As a result, your components behave according to the desired logic as you make modifications.

Once you finalize a packaged component, you can no longer move it with the Package functionality; however, you can modify or redefine the placement using the Component Placement dialog box.

## About Merge by Reference

The Merge by Reference feature creates an external reference between two Pro/Engineer objects. In this type of relationship the merged part is dependent on information in the referenced part. The merge functionality can be accessed by **Feature > Create > Data Sharing** to open the DATA SHARING menu or **Insert > Shared Data**. The Merge and copy features have been added to the Data Sharing group to allow the user to clearly specify the parent model, and merged model. The parent model will be the active working model. This transition has also converted the Merge and copy features to a standard feature creation interface. Beginning with release 2001, you can include names of datum entities from the reference model in the merged model. Layers and colors are automatically included.

## Using the Component Placement Dialog Box



A toolbar across the top of the dialog box contains eight buttons that allow access to the following functions:

- Show the component in a separate window while specifying its constraints.
- Show the component in the assembly window while specifying its constraints.
- Create an interface.
- Retrieve references that are not available in the current simplified representation. This option is available when you redefine a component in a simplified representation and that component depends on components not available in the simp rep.
- Preferences.

The Constraints section of the dialog box contains a three-column layout as follows:

- Constraint type.
- Offset.
- Checkbox indicating if the constraint is active or inactive.

Buttons below the constraints list allow you to Add, Remove and Flip the selected constraint. Two buttons

specify the  Default and  Fix constraints. These are considered special constraint types because they do not require picking any component or assembly references.

The Component Reference section displays the selected component and assembly references.

## About Creating Components in Assembly Mode

Using the Component Create dialog box, you can create different types of components: parts, subassemblies,

skeleton models, and bulk items.

**Note:** The Advanced Pro/ASSEMBLY Extension license is required for creating skeleton models. Online documentation for Advanced Pro/ASSEMBLY Extension provides detailed information.

The following methods allow component creation in the context of an assembly without requiring external dependencies on the assembly geometry:

- Create a component by copying another component or existing start part or start assembly
- Create a component with default datums
- Create an empty component

You can also perform the following operations:

- Create the first feature of a new part; this initial feature is dependent on the assembly
- Create a part from an intersection of existing components
- Create a mirror copy of an existing part or subassembly

**Note:** You cannot reroute components created in Assembly mode

## To Create a Solid Part by Copying From an Existing Part

You can specify a component to copy "on-the-fly" and place the copy in the assembly immediately.

**Note:** A part that contains a Shrinkwrap feature is associative and, therefore, cannot be used as the source for creating a new part using **Copy From Existing**.

1. Choose **ASSEMBLY > Component > Create**, or choose **Component > Create** from the pop-up menu in the Model Tree window. The Component Create dialog box appears.
2. Click **Part**, and then **Solid**.
3. Accept the default name or enter a new name, and click **OK**. The Creation Options dialog box opens.
4. Click **Copy From Existing**.
5. Click **Browse**, select the name of a component to copy, and click **Open**. The name of the selected component appears in the **Copy From** text box.
6. You can select **Leave Component Unplaced** to include the new component in the assembly without defining placement constraints.
7. Click **OK**.

The new part is placed in the assembly, or it is included in the assembly as an unplaced component if you selected **Leave Component Unplaced**.

## Copying Parts with Layouts or External References

If you create a part by copying from another part that has a layout declared to it, you must confirm that you would like to declare that layout to the newly created copied part as well. Otherwise, the system does not copy the layout declarations to the newly created part.

If you try to create a part from an existing part with external references that *are not* locally backed up, the system aborts the copy. If you try to create a part from an existing part that has external references that *are* locally backed up, the system will notify you that those external references will be made permanently independent in the new component, and ask whether you wish to continue.

## Tip: Set the Start Model Directory Path

Set the `start_model_dir` configuration file option to specify a complete path to the directory where start

parts and assemblies are stored. Then, when you browse the directory structure to select a start component to copy from, the File Open dialog box will look in this directory by default.

## To Create a Solid Part and Set Default Datums

You can create a component and assemble it automatically to references in the assembly. The system creates constraints to locate the default datum planes of the new component relative to the selected assembly references.

1. Choose **ASSEMBLY > Component > Create**, or choose **Component > Create** from the pop-up menu in the Model Tree window. The Component Create dialog box appears.
2. Click **Part**, and then **Solid**.
3. Accept the default name or enter a new name, and click **OK**. The Creation Options dialog box opens.
4. Click **Locate Default Datums**. The **Locate Datums Method** area of the dialog box opens.
5. Select references from the assembly using one of the **Locate Datums Method** options:
  - **Three Planes**
  - **Axis Normal To Plane**
  - **Align Csys To Csys**

Click *See Also* for detailed information.

The system creates a new part with default datums and places you in feature creation mode.

6. Define features for the new part that will automatically use the default datum planes for their references.
  - If you used either the **Three Planes** or the **Axis Normal To Plane** option, the sketching plane is the first plane that you selected.
  - If you used the **Align Csys To Csys** option, you must select the sketching plane.

Once you create a feature or quit its creation, the system places the new component in the assembly in the way in which its default planes are mated (by **Mate Offset** with zero offsets) to the selected references in the assembly. In the case of **Axis Normal To Plane**, the system also aligns the component axis with the selected assembly axis.

You can use the **Mod Dim** command on the ASSEM MOD menu to modify the value of the offsets in the placement constraints, or redefine the component placement constraints entirely.

## Locate Datums Methods

Select references from the assembly using one of the following methods:

- Click **Three Planes** and **OK**, and select three orthogonal datum planes from the assembly to which the default datum planes of the newly created component will be assembled.
- Click **Axis Normal To Plane** and **OK**, and select a single datum plane and an axis that is normal to it. The system then creates a new component with a datum plane and an axis which it uses to place the new component with respect to the rest of the assembly.
- Click **Align Csys To Csys** and **OK**, and select a coordinate system in the top-level assembly. The system then creates a new component with a default coordinate system and default datum planes which it uses to place the new component relative to the rest of the assembly.

## To Create an Empty Part

You can create a part with no initial geometry.

1. Choose **ASSEMBLY > Component > Create**, or choose **Component > Create** from the pop-up menu in the Model Tree window. The Component Create dialog box appears.
2. Click **Part**, and then **Solid**.

3. Accept the default name or enter a new name, and click **OK**. The Creation Options dialog box opens.
4. Click **Empty**.
5. You can select **Leave Component Unplaced** to include the new part in the assembly without defining placement constraints.
6. Click **OK**.

The new part is placed in the assembly, or it is included in the assembly as an unplaced component if you selected **Leave Component Unplaced**.

## Empty Components in an Assembly

You can create a part or subassembly without any geometry and it appears in the Model Tree window. If you save the top-level assembly, the system saves this empty component also.

For empty parts or assemblies with no geometry, the Model Size parameter has a value of zero.

When you create a feature in an empty component in an empty assembly, you are put directly into Sketcher mode and are not prompted for sketcher references (or external references, in the case of datum geometry).

When you define empty components within the context of an assembly, the system places them in the default location of their parent assemblies (using the **Default** constraint type).

Once you have created an empty component within an assembly, you can create default datum planes by retrieving the empty part or subassembly into a separate window and choosing **Feature, Create, Datum, and Plane**. You can also fill these components.

**Note:** When you fill an empty component with features that reference the existing assembly, you can no longer redefine the placement of the component.

## To Create a Solid Part and Its First Feature

You can create the first feature of a new part. This initial feature is dependent on the assembly.

1. Choose **ASSEMBLY > Component > Create**, or choose **Component > Create** from the pop-up menu in the Model Tree window. The Component Create dialog box appears.
2. Click **Part**, and then **Solid**.
3. Accept the default name or enter a new name, and click **OK**. The Creation Options dialog box opens.
4. Click **Create First Feature**, and click **OK**. The FEAT CLASS menu appears.
5. Select the class of feature that you want to create.
6. Create the geometry of the new part either by referencing existing geometry within the assembly or without using references:
  - Specify a sketching plane and a sketching reference from the existing geometry of the assembly to create the geometry of the new part.  
**Note:** The newly created component will have external dependencies to the assembly, and therefore you will not be able to redefine its placement.
  - Or—if no geometry yet exists in the assembly—you can create the geometry of the new part without using references.
7. Click **OK**.

## To Create a Part from an Intersection

You can create a part in Assembly mode by intersecting several existing components. These parts do not need to have the same units of measure. You can also modify existing parts in an assembly by intersecting them with another part.

1. Choose **ASSEMBLY > Component > Create**, or choose **Component > Create** from the pop-up menu in the

- Model Tree window. The Component Create dialog box appears.
2. Click **Part**, and then **Intersect**.
  3. Accept the default name or enter a new name, and click **OK**.
  4. Select parts to intersect.
- The new part represents the common volume of selected components.

## Parts Created from an Intersection

When working with a part created from an intersection, keep in mind the following:

- You cannot move the resulting part.
- You cannot use harness parts to create a part by intersection.
- You cannot retrieve the part until the assembly to which it belongs is in memory.

## To Trim a Part in an Assembly to a Common Volume

1. Choose **Modify > Mod Part**, then select one part in the assembly.
2. Choose **Feature > Create > Solid > Intersect**.
3. Select a second part with which to intersect the first part (you can access this option *only* in Assembly mode).

## To Create a Mirror Copy of a Part

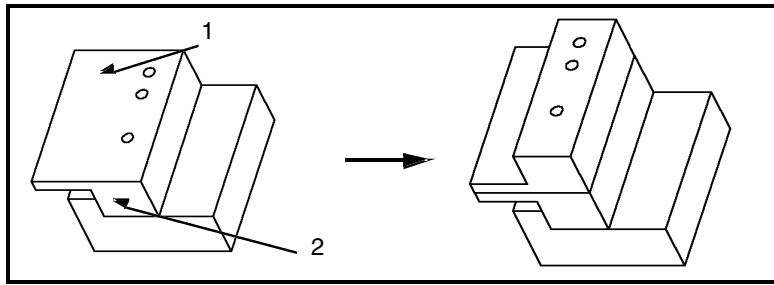
1. Choose **ASSEMBLY > Component > Create**, or choose **Component > Create** from the pop-up menu in the Model Tree window. The Component Create dialog box opens.
  2. Click **Part**, and then **Mirror**.
  3. Enter a name for the new part. The MIRROR PART menu appears.
  4. Choose one of the following:
    - **Reference**—References the second part to obtain its information. When the referenced part changes, the mirrored part changes.
    - **Copy**—Copies all the features and relations of the original part into the mirrored part. The mirrored part then becomes a separate, unrelated object.
  5. Select a part in the assembly to mirror.
  6. Select or create the mirror plane.
- Note:** It is good practice to mirror about a plane belonging to the part to be mirrored.

## Referenced and Copied Mirrored Parts

You can regenerate the mirrored (new) part in Part mode and place it into other assemblies. However, if you used the **Reference** option to mirror the part, the system always retrieves the part from which it was created (if you used **Copy**, it does not retrieve the original part). If the part references another component, or if it uses an assembly datum, the system also retrieves the assembly and any referenced components.

## Example: Creating a Mirror Copy of a Part

The following figure shows selecting a part to mirror and a mirror plane, and the new mirrored part.



- 1 Mirror plane
- 2 Select this part

## To Create a Subassembly by Copying an Assembly

You can specify a component to copy "on-the-fly" and place the copy in the assembly immediately.

1. Choose **ASSEMBLY > Component > Create**, or choose **Component > Create** from the pop-up menu in the Model Tree window. The Component Create dialog box appears.
2. Click **Subassembly**, and then **Standard**.
3. Accept the default name or enter a new name, and click **OK**. The Creation Options dialog box opens.
4. Click **Copy From Existing**.
5. Click **Browse**, select the name of a component to copy, and click **Open**. The name of the selected component appears in the **Copy From** text box.
6. You can select **Leave Component Unplaced** to include the new component in the assembly without defining placement constraints.
7. Click **OK**.

The new subassembly is placed in the assembly, or it is included in the assembly as an unplaced component if you selected **Leave Component Unplaced**.

## To Create a Subassembly and Set Default Datums

You can create a subassembly and assemble it automatically to references in the assembly. The system creates constraints to locate the default datum planes of the new subassembly relative to the selected assembly references.

1. Choose **ASSEMBLY > Component > Create**, or choose **Component > Create** from the pop-up menu in the Model Tree window. The Component Create dialog box appears.
2. Click **Subassembly**, and then **Standard**.
3. Accept the default name or enter a new name, and click **OK**. The Creation Options dialog box opens.
4. Click **Locate Default Datums**. The **Locate Datums Method** area of the dialog box opens.
5. Select references from the assembly using one of the **Locate Datums Method** options:
  - **Three Planes**
  - **Axis Normal To Plane**
  - **Align Csys To Csys**

Click *See Also* for detailed information.

The system creates a new subassembly with default datums and places you in feature creation mode.

6. Define features for the new subassembly that will automatically use the default datum planes for their references.
  - If you used either the **Three Planes** or the **Axis Normal To Plane** option, the sketching plane is the first plane that you selected.
  - If you used the **Align Csys To Csys** option, you must select the sketching plane.

Once you create a feature or quit its creation, the system places the new component in the assembly in the way



in which its default planes are mated (by **Mate Offset** with zero offsets) to the selected references in the assembly. In the case of **Axis Normal To Plane**, the system also aligns the component axis with the selected assembly axis.

You can use the **Mod Dim** command on the ASSEMBLY MOD menu to modify the value of the offsets in the placement constraints, or redefine the component placement constraints entirely.

## To Create an Empty Subassembly

You can create a subassembly with no initial geometry.

1. Choose **ASSEMBLY > Component > Create.**, or choose **Component > Create** from the pop-up menu in the Model Tree window. The Component Create dialog box appears.
2. Click **Subassembly**, and then **Standard**.
3. Accept the default name or enter a new name, and click **OK**. The Creation Options dialog box opens.
4. Click **Empty**.
5. You can select **Leave Component Unplaced** to include the new subassembly in the assembly without defining placement constraints.
6. Click **OK**.

The new subassembly is placed in the assembly, or it is included in the assembly as an unplaced component if you selected **Leave Component Unplaced**.

## Empty Components in an Assembly

You can create a part or subassembly without any geometry and it appears in the Model Tree window. If you save the top-level assembly, the system saves this empty component also.

When you create a feature in an empty component in an empty assembly, you are put directly into Sketcher mode and are not prompted for sketcher references (or external references, in the case of datum geometry).

When you define empty components within the context of an assembly, the system places them in the default location of their parent assemblies (using the **Default** constraint type).

Once you have created an empty component within an assembly, you can create default datum planes by retrieving the empty part or subassembly into a separate window and choosing **Feature, Create, Datum**, and **Plane**. You can also fill these components.

**Note:** When you fill an empty component with features that reference the existing assembly, you can no longer redefine the placement of the component.

For empty parts or assemblies with no geometry, the Model Size parameter has a value of zero.

## To Create a Mirror Copy of a Subassembly

1. Retrieve an assembly that contains a subassembly to be mirrored, and choose **ASSEMBLY > Component > Create**, or choose **Component > Create** from the pop-up menu in the Model Tree window. The Component Create dialog box opens.
2. Click **Subassembly**, and then **Mirror**.
3. Enter a name for the new subassembly to be created, and click **OK**. The Mirror Subassembly dialog box opens, and the GET SELECT menu appears.
4. Select a subassembly to mirror. The name of the selected subassembly is displayed in the Subassembly Reference area of the dialog box.

**Note:** You can select any fully placed subassembly of the current assembly to mirror, including a previously mirrored mirror, or an empty or packaged subassembly.

5. To select the subassembly reference, click the selection button and use the **GET SELECT** menu.

6. To select or create the planar reference (the mirror plane), click the selection button and use the SETUP PLANE menu to select a plane or create a datum about which to mirror. You can use any planar reference, as is the case when creating a mirrored part in Part mode. It is good practice to mirror about a plane belonging to the part to be mirrored.
7. Click **OK**. The **Mirror Subassembly Components** dialog box opens.  
This dialog box provides a method for renaming target components, including:
  - A tree-like view showing the hierarchy of the mirrored subassembly
  - The ability to select components to skip during mirroring
  - The ability to select components to leave unmirrored, thus reusing the original model
8. Click **OK**. The system creates the mirrored subassembly.

## Mirror Copies of Subassemblies

The **Mirror** option, available when creating subassemblies using the Create Component dialog box, provides an automated method for creating a mirror copy of a subassembly, eliminating the laborious process of mirroring each component manually. Subassembly mirrors are useful for assemblies that have a symmetric mirror design.

You can create subassembly mirrors for both symmetric and asymmetric components. The mirrored components are generated as a mirror reference, as when individual parts are mirrored manually. To create a mirror, you specify the plane of symmetry by identifying an existing datum plane or by creating one.

When a subassembly is mirrored, the assembly features that belong to that subassembly are mirrored as well. Additionally, layer information from the original models are copied to the new layers. These layers are independent after they are copied.

The system treats a mirrored subassembly in the same way as a subassembly, and the Merge feature is the first feature.

When a component is mirrored individually, or when a subassembly is mirrored, the new component is placed with a **Default** constraint. The Bill of Materials is a snapshot (independent), and the geometry is By Reference (dependent).

You can redefine either the placement of the newly created subassembly, or the placement of the components that make up this newly created subassembly.

Redefining has the following results:

- The placement of the newly created subassembly mirror (or the newly created components) is independent of any changes to the placement of the parent (mirrored) components.
- The new subassembly mirror will update according to any subsequent geometric changes to the parent components.

You can mirror packaged and unplaced components and redefine placement (making the mirrors independent of changes to placement of the parent). Mirrors of packaged components are frozen in space where they are and placed by default. If the parent assembly is outside the active subassembly, the new mirrored subassembly remains frozen if the active subassembly is subsequently loaded without the top-level assembly in session.

If the parent subassembly has any components excluded, substituted, or in a graphics representation state because of a simplified representation setting, these components are excluded because the master assembly is not in session. The system displays a message, "Only components of the subassembly which are in a Master or Geometry Rep state will be mirrored."

The following additional rules apply to subassembly mirrors:

- During a copy, once a subcomponent has been mirrored once, the mirror is used for later occurrences (for performance).
- Mirrored assemblies can be reused in other parent assemblies.
- You can control exploding of mirrored assemblies.

- A mirror copy of a mirror subassembly behaves the same way as a mirror of a mirrored part.
- You can mirror packaged components and redefine placement (making the mirrors independent of changes to placement of the source).
- Skeleton models are renamed but are still recognized as skeletons.
- Bulk items are copied.
- Layers are copied.
- Patterned components create a mirror pattern set in the mirrored assembly.
- Assembly features that reside outside the subassembly are ignored.
- Assembly features within the subassembly being mirrored are copied and the new Assembly cut features can have their component intersections modified.

## About Working with Assembly Components

Pro/ENGINEER treats components in an assembly in much the same way that it treats features in a part. Therefore, you can use the commands in the COMPONENT menu in exactly the same way in Assembly mode as in Part mode.

You can use the **Adv Utils** command to access advanced assembly component functionality.

**Note:** The COMPONENT menu commands can also be accessed from the pop-up menu in the Model Tree window.

## To Delete a Component from an Assembly

Choose COMPONENT > **Delete** to delete a component and its children from an assembly.

## To Suppress a Component from Active Memory

Choose COMPONENT > **Suppress** to suppress a component from active memory.

## Suppressed Components

When you retrieve an assembly, the system does not retrieve suppressed components at all. This saves time and memory when working with large assemblies. Suppressed components do not appear in mass properties and cross sections, and you cannot store them (when using the **Save As** or **Backup** option).

## To Freeze Suppressed Children

When you suppress an assembly feature or component that has children, the CHILD menu appears with the **Freeze** command available for the highlighted child. The system always locates frozen components at the old placement until it can regenerate them successfully, either by resuming the parent, redefining the placement, or rerouting the child. The CHILD menu appears for each child of the suppressed feature/component.

## To Resume Components and Assembly Features

Choose COMPONENT > **Resume** to resume suppressed components and also assembly features.

## To Reroute Placement References

Choose **COMPONENT > Reroute** to reassign the placement references of a component. The procedure is the same as for rerouting features in Part mode.

**Note:** You cannot reroute components created in Assembly mode

## To Reorder Components in an Assembly

Choose **COMPONENT > Reorder** to reorder the occurrence of a component in an assembly. The procedure is the same as for reordering features in Part mode.

## To Insert a Component in the Regeneration List

Choose **COMPONENT > Insert Mode** to place components at an earlier location in the component regeneration list (to insert one component before another).

## To Delete Pattern Member Components

Click **COMPONENT > Del Pattern** to remove all instances of a patterned component except the pattern leader, and remove pattern definition.

## To Create a Group of Components and Features

Click **COMPONENT > Adv Utils > Group** to create a user-defined group of components and features.

You can use the **Group** command as in Part mode. However, there are some restrictions for using this command in Assembly mode.

## To Create User-Defined Features (UDFs)

1. Click **COMPONENT > Adv Utils > UDF Library** to open the UDF menu. The UDF menu has the following commands:
  - **Create**—Add a new UDF to the UDF Library.
  - **Modify**—Modify an existing UDF.
  - **List**—List all the UDFs in the current directory.
  - **Dbms**—Perform database management functions for the current UDF.
  - **Integrate**—Resolve the differences between the source and the target UDFs.
2. Enter a name for the UDF.
3. Choose one of the following options from the UDF OPTIONS menu, followed by **Done**:
  - **Stand Alone**—Pro/ENGINEER copies all the required information to the UDF.
  - **Subordinate**—Pro/ENGINEER copies most of the information from the original part at run time.
4. The system displays the UDF feature creation dialog box with the following elements:
  - **Features**—(Defining) Select features or components to include in the UDF.
  - **Assoc Models**—(Optional) Select component models to include in the UDF (available only if you select **Subordinate** above). These can include models created in the assembly. If the model has external references, the system will prompt for them. When the parts selected are created in the assembly, the newly placed UDF will create a new part with a new name.

- **Ref Prompts**—(Required) Enter prompts for specifying placement references. The system prints these prompts to guide you when you place the UDF.
  - **Var Elements**—(Optional) Specify feature elements that you want to be able to redefine when you place the UDF in an assembly.
  - **Var Dims**—(Optional) Select dimensions that you want to modify when you place the UDF in an assembly.
  - **Family Table**—(Optional) Create a Family Table of UDFs.
  - **Pro/Program**—(Optional) Defining this element opens the PROGRAM menu with three commands available: **Show Design**, **Edit Design**, and **J-Link**.
  - **Ext Symbol**—(Optional) Include external dimensions and parameters in the UDF.
5. Define all the elements to include in the UDF.
  6. Click **OK**.

## About Redefining Placement Constraints

After you place a component, you can redefine its placement constraints. Using the Component Placement dialog box, you can add or remove constraints for the active component, and redefine any of its constraints in the following ways:

- Change the following constraint types to the others in this list:
  - **Align**
  - **Mate**
  - **Orient**
- Reset an **Align** constraint to forced or unforced
- Flip sides
- Modify the offset value
- Specify new assembly references
- Specify new component references
- Switch between allowing and disallowing system assumptions

When redefining component placement, you can select datums or make them on the fly, as is the case when you are placing a component.

When you use the **Default** constraint, the system places the component at the assembly origin.

Use the **Fix** constraint to fix the current location of the component that was moved or packaged. The system fully constrains the packaged component in its current location.

You can redefine components in a simplified representation that have not been substituted or excluded in the current representation.

## To Redefine Component Constraints

1. Choose **ASSEMBLY > Component > Redefine** and then select the component to redefine, or right-click the assembly or component name in the Model Tree and choose **Redefine** from the pop-up menu. The Component Placement dialog box opens.
2. Select one of the constraints listed in the Constraints area of the dialog box. For each constraint, you can choose what to redefine using the following options:
  - **Constraint Type**—Changes the constraint type to any reasonable type for the assembly.  
**Note:** You can only change the type for the following constraints: **Align**, **Mate**, and **Orient** (you can change any of these constraints to any of the others in this list).
  - **Assembly Reference**—Specifies a new assembly reference (for example, changes the surface on the assembly to which you are going to align the component).

- **Component Reference**—Specifies a new component reference for a placed component (for example, changes which surface on the component you are going to align with the assembly).
3. You can select **Remove** or **Add** at any time:
- To delete a placement constraint for the component, select one of the constraints listed in the Constraints area, and then click **Remove**.  
The system deletes the selected constraint from the current list and updates the message in the Placement Status area. You may need to add a new constraint.
  - The **Retr Refs** option appears when you redefine a component in a simplified representation, and that component depends on components that are not in the simplified representation. Click **Retr Refs** to retrieve any other components that define the location of the component.
  - To add a new constraint to the current list, click **Add**. Select a type of constraint from the **Constraint Type** list. Select a reference on the component and a reference on the assembly, in either order, to define a placement constraint. Click *See Also* for detailed information.
4. After you redefine the constraints for the active component, click **OK**.
- Note:** You can also use the Move options, but if the component has children, you must fully constrain the component before exiting the interface.

## About Replacing Assembly Components

When a component is replaced, the system swaps the new component into the assembly in the same place geometrically, and into the Model Tree.

You can automatically replace an assembly component with another model that shares similar constraints and references, if the model to be replaced is a shrinkwrap, a member of a family table or a functional interchange assembly, or if it is declared to a layout.

These replacement options are available in the Replace Component dialog box when applicable to the component selected for replacement. Unavailable options appear dimmed.

The system automatically assembles the new component, if possible. If necessary, for example, if references are missing, the Component Placement dialog box opens, and you must manually establish placement constraints.

When you replace a component with one that is unrelated by family table, interchange assembly, or layout, you must manually reposition the replacement component and any components that were assembled to the original part. The system always tries to assemble the component automatically. If it is unable to do so, it keeps as many constraints as possible.

You can also replace a component by new copy, that is, by creating a new component based on the existing component and swapping the new component into the assembly. This replacement method is especially useful for creating a new skeleton model and swapping it into the assembly.

You can replace a skeleton model only by family table and by new copy.

You can replace more than one component at a time in an assembly. Either multiple occurrences of a single component can be replaced by a single component, or multiple components can be replaced by multiple unique components.

**Note:** Substituting components is not to be confused with replacing components. Substitution is performed to exchange one component in a simplified representation for another and is performed in the context of a simplified representation. Online Help documentation for Advanced Pro/ASSEMBLY Extension provides detailed information.

Click *See Also* for detailed information about component replacement procedures.

## To Replace a Component by Family Table Member

This method of replacement allows you to replace a generic model with a family table member. You can replace

components in an assembly using family table definitions associated to it. Using a family table, you can make different instances in a part family table automatically interchangeable by creating them from the same original part, the generic instance. The system automatically places the new component the same way it placed the original component because each instance contains the same references.

**Note:** You can create a skeleton model as the replacement model.

1. Retrieve an assembly, choose **Component > Adv Utils > Replace**, and then select one or more components to be replaced; or select one or more components from the Model Tree, and choose **Replace** from the pop-up menu. The REPLACE COMP, SELECT MDL, and GET SELECT menus appear. The Replace Component dialog box opens. An empty Component Replace column is displayed in the Model Tree.  
You can select a component from the screen, or use the SELECT MDL menu and then select or designate one or more components of the assembly to be replaced. Click *See Also* for detailed information.
2. Click **By Family Table Member**.
3. Click **Use Geom Only Rep.** if you want the replacement model to be a geometry representation.
4. Click **Mark as a Skeleton Model** if you want the newly created component to be a skeleton model.  
After you select **Mark as a Skeleton Model**, this option appears dimmed and you cannot clear it.
5. Click **Browse**, select the replacement model, and click **OK**. (You cannot enter a replacement model name manually.) The name of the selected component appears in the text box next to **Browse**.  
You can click **Clear** to clear the name and browse again for the replacement model.
6. Click **Apply**. The name of the replacement model appears in the Component Replace column in the Model Tree next to the name of the component being replaced.
7. Modify the list of components to be replaced:
  - Use the SELECT MDL and GET SELECT menus to select additional components to replace.
  - Use the REPLACE COMP menu to add or remove items from the list displayed in the Component Replace column in the Model Tree.
8. Choose **Done** from the **REPLACE COMP** menu. The system replaces the original component and automatically assembles the new component. The system swaps the new component into the assembly in the same place geometrically, and into the Model Tree.  
If you selected **Mark as a Skeleton Model**, the newly created component is a skeleton model.  
**Note:** You can automatically replace models only if the references assembled within the replaced model are also present in the new replacement model. If they are not, the Component Placement dialog box opens, and you must select references to replace the missing ones, and click **OK**.
9. You can continue to replace additional components.
10. Choose **Done/Return** from the ADV COMP UTL menu and **Done/Return** from the COMPONENT menu.

## Retrieving the Replacement Model References

Click **Retrieve Refs** (selected by default) if you want to retrieve the replacement model references. This option is available when you replace a component in a simplified representation and that component depends on components that are not in the simplified representation. Click **Retrieve Refs** to retrieve any other components that define the location of the component.

## Adding or Removing Components from the Component Replace List

Using the REPLACE COMP menu, you can add or remove items from the replacement list displayed in the Component Replace column in the Model Tree. After choosing one of the following commands, use the SELECT MDL menu to select components.

- **Add**—Selects components to be replaced.
- **Remove**—Selects components to be taken off the replacement list.
- **Undo Last**—Cancels the last previous addition or removal of a component.

- **Info**—Selects a component about which to obtain information.

## Suspending External References in the Replacement Model

When you replace a component with external references created in the assembly in which it is being replaced, the system displays the following message and a confirmation prompt, "The component being replaced has external references, which will exist in the newly created copy. They will be suspended in the replaced model."

- Choose **Confirm** to continue with the operation and suspend the references in the model which was removed from the assembly.
- Choose **Cancel** to cancel the replace operation and return to the REPLACE COMP menu.

## Replacing Additional Components

After you apply a component replacement, you can select the next model to be replaced:

- When you select the next model to be replaced from the screen, you must click **Apply** after each replacement.
- When you select the next model to be replaced from the Model Tree, you can select multiple models to be replaced.

**Note:** If you select additional models to be replaced, and the previously selected replacing model can be used to replace them, it will be applied by default. You also can specify a new replacing model.

## To Replace a Component by Interchange Assembly

This method of replacement allows you to replace a component in an assembly with a component that is a member of a functional interchange assembly. When you swap a component by using an interchange assembly, you preserve the parent/child relationships between the components. You can automatically replace a component with another component because you have defined references for the parents and children of the component.

1. Retrieve an assembly, choose **Component > Adv Utils > Replace**, and then select one or more components to be replaced; or select one or more components from the Model Tree, and choose **Replace** from the pop-up menu. The REPLACE COMP, SELECT MDL, and GET SELECT menus appear. The Replace Component dialog box opens. An empty Component Replace column is displayed in the Model Tree. You can select a component from the screen, or use the SELECT MDL menu and then select or designate one or more components of the assembly to be replaced. Click *See Also* for detailed information.
2. Click **By Interchange Assembly**.
3. Click **Use Geom Only Rep.** if you want the replacement model to be a geometry representation.
4. Click **Browse**, select and expand the interchange assembly, select the replacement model, and click **OK**. (You cannot enter a replacement model name manually.) The name of the selected component appears in the text box next to **Browse**. You can click **Clear** to clear the name and browse again for the replacement model.
5. Click **Apply**. The name of the replacement model appears in the Component Replace column in the Model Tree next to the name of the component being replaced.
6. Modify the list of components to be replaced:
  - Use the SELECT MDL and GET SELECT menus to select additional components to replace.
  - Use the REPLACE COMP menu to add or remove items from the list displayed in the Component Replace column in the Model Tree.
7. Choose **Done** from the REPLACE COMP menu. The system replaces the original component and automatically assembles the new component. The system swaps the new component into the assembly in the



same place geometrically, and into the Model Tree.

**Note:** You can automatically replace models only if the references assembled within the replaced model are also present in the new replacement model. If they are not, the Component Placement dialog box opens, and you must select references to replace the missing ones, and click **OK**.

8. You can continue to replace additional components.
9. Choose **Done/Return** from the ADV COMP UTL menu and **Done/Return** from the COMPONENT menu.

## To Replace a Component by Layout

You can use a layout to replace one component with another automatically. This method of replacement allows you to replace the model with a component that has declared references. Automatic assembly is available if you have set up layout declarations for the placement references. When replacing a component that has children of its own, through a layout, the system does not automatically reassemble its children.

1. Retrieve an assembly, choose **Component > Adv Utils > Replace**, and then select one or more components to be replaced; or select one or more components from the Model Tree, and choose Replace from the pop-up menu. The REPLACE COMP, SELECT MDL, and GET SELECT menus appear. The Replace Component dialog box opens. An empty Component Replace column is displayed in the Model Tree.  
You can select a component from the screen, or use the SELECT MDL menu and then select or designate one or more components of the assembly to be replaced. Click *See Also* for detailed information.
2. Click **By Layout**.
3. Click **Use Geom Only Rep.** if you want the replacement model to be a geometry representation.
4. Click **Browse**, select the replacement model, and click **OK**. (You cannot enter a replacement model name manually.) The name of the selected component appears in the text box next to **Browse**.  
You can click **Clear** to clear the name and browse again for the replacement model.
5. Click **Apply**. The name of the replacement model appears in the Component Replace column in the Model Tree next to the name of the component being replaced.
6. Modify the list of components to be replaced:
  - Use the SELECT MDL and GET SELECT menus to select additional components to replace.
  - Use the REPLACE COMP menu to add or remove items from the list displayed in the Component Replace column in the Model Tree.
7. Choose **Done** from the REPLACE COMP menu. The system replaces the original component and automatically assembles the new component if you have set up layout declarations. The system swaps the new component into the assembly in the same place geometrically, and into the Model Tree. If you have not set up layout declarations, the Component Placement dialog box opens, and you must select references to replace the missing ones, and click **OK**.
8. You can continue to replace additional components.
9. Choose **Done/Return** from the ADV COMP UTL menu and **Done/Return** from the COMPONENT menu.

## Layout Replacement

You can use layouts to replace components within an assembly and to place the replacement models automatically. A layout is an automatic method of interchanging components. The system assembles the new replacement model into the same position as the original component and using the same constraints as for the original component, and you do not have to specify assembly constraints manually.

To place components automatically using a layout, you must first define global placement references in the layout, and then identify the corresponding global placement references on the components themselves.

Using layouts for automatic assembly, you can create interchangeability. If you assign the same global placement reference names to references in multiple components (parts or assemblies), you can assemble any one of those multiple components that have been declared to the layout, and the system places the component automatically. If the system can automatically place multiple components, you can use any component with global placement references to replace automatically any other component with the same global placement

references.

## To Replace a Component by New Copy

This method of replacement allows you to replace a model with a newly created copy of the component. You cannot replace multiple models. You can use this method to replace only a single part model component; you cannot use this method to replace a subassembly.

When you replace a family member (generic or instance) by new copy, the system does not copy family table information from the replaced model, just as in **Copy From**.

**Note:** You can create a skeleton model as the replacement model if the following conditions are met: if a skeleton component is allowed at this place in the assembly structure, that is, if the component being replaced is the first component (without even assembly features before it); and if `multiple_skeletons_allowed` is set to yes (allowing multiple skeletons); and if the component being replaced comes just after an existing skeleton component in the assembly tree.

1. Retrieve an assembly, choose **Component > Adv Utils > Replace**, and then select one component to be replaced; or select one component from the Model Tree, and choose **Replace** from the pop-up menu. The REPLACE COMP, SELECT MDL, and GET SELECT menus appear. The Replace Component dialog box opens. An empty Component Replace column is displayed in the Model Tree.  
You can select a component from the screen, or use the SELECT MDL menu and then select or designate one component part model of the assembly to be replaced. Click *See Also* for detailed information.
2. Click **By New Copy**.
3. Click **Use Geom Only Rep.** if you want the replacement model to be a geometry representation.
4. Click **Mark as a Skeleton Model** if you want the newly created component to be a skeleton model. This option is available only if the skeleton model can be put at this location.  
After you select **Mark as a Skeleton Model**, this option appears dimmed and you cannot clear it.
5. Accept the default name, or enter a name for the new model in the **Name** box, and click **Apply**. The name of the replacement model appears in the Component Replace column in the Model Tree next to the name of the component being replaced.
6. You can remove the item listed to be replaced and replace it with another:
7. Modify the listing of the component to be replaced:
  - Use the SELECT MDL and GET SELECT menus to select a component to replace.
  - Use the REPLACE COMP menu to add or remove an item from the list displayed in the Component Replace column in the Model Tree.
8. Choose **Done** from the REPLACE COMP menu. The system creates a new model, replaces the original component with the new model, and automatically assembles the new component. The system swaps the new component into the assembly in the same place geometrically, and into the Model Tree.
9. If you selected **Mark as a Skeleton Model**, the newly created component is a skeleton model.
10. You can continue to replace additional components.
11. Choose **Done/Return** from the ADV COMP UTL menu and **Done/Return** from the COMPONENT menu.

## Copied Replacement Components

The default file name PRT000X is assigned to the new component. You can change the name.

If you create a skeleton model, the default file name `assemblyname_skel` is used for the first skeleton model in an assembly. Subsequently created skeleton models are named `assemblyname_skel0002`, `assemblyname_skel0003`, and so forth.

The following rules and restrictions apply to components created using the **By New Copy** option:

- You cannot replace multiple models.
- You can replace a single part, not a subassembly.
- The copy is completely independent. There is no dependency back to the replaced model. The operation is

analogous to performing a **Save As** or **Copy From** operation.

- All attributes of the part being copied are copied into the new component, including the following:
  - All features, including suppressed features
  - Colors set at the part level
  - Layers, and layer settings and assignments
- A model with external copy geometry features can be copied. The newly created model has an external reference to the same model to which the original model has a reference.
- Models that reference assembly features can be copied.
- If the part that is copied is declared to a layout, the newly created part is also declared to the layout.
- If the part that is copied has relations, the new part has relations.
- You can copy a part model component into a new skeleton model. You can generate a native skeleton model, based on a native part model, and have it replace the part model in an assembly, with all references remapped to the new skeleton model. This effectively allows a part to be designated as a native skeleton model, through the use of a new model file.
- A component can become a skeleton model only if it does not violate any accepted skeleton model behavior; for example, it cannot have simplified representations. In addition, the component must be either the first component in the assembly, or the first nonskeleton component in the assembly.
- A skeleton model can be copied only as a skeleton model, not as a regular part.

## Tip: Allow Multiple Skeleton Replacement Copies

When replacing a component By New Copy, and a skeleton model can be created as the replacing part, the configuration file option `multiple_skeletons_allowed` determines whether multiple skeletons can be created as replacing parts:

- When set to `no`, only the first component (not a pattern leader) is allowed to become a skeleton model.
- When set to `yes`, the first component after the existing skeleton group is allowed to become replaced by a copy of itself, which will be a skeleton model (pattern leader is allowed).

## To Replace a Component Manually

This replacement method replaces the model with a component you select and positions that component manually. Although manual replacement is similar to deleting and assembling a new component, manual replacement allows you to automatically place the newly replaced component in the regeneration position formerly occupied by the old component.

**Note:** Unlike the deletion process, Pro/ENGINEER does not highlight any children when you use the manual replacement technique. Therefore, you may want to investigate the parent/child references of a component before you replace it.

1. Retrieve an assembly, choose **Component > Adv Utils > Replace**, and then select one or more components to be replaced; or select one or more components from the Model Tree, and choose **Replace** from the pop-up menu. The **REPLACE COMP**, **SELECT MDL**, and **GET SELECT** menus appear. The Replace Component dialog box opens. An empty Component Replace column is displayed in the Model Tree. You can select a component from the screen, or use the **SELECT MDL** menu and then select or designate one or more components of the assembly to be replaced. Click *See Also* for detailed information.
2. Click **Manually**.
3. Click **Use Geom Only Rep.** if you want the replacement model to be a geometry representation.
4. Click **Browse**, select the replacement model, and click **Open**. (You cannot enter a replacement model name manually.) The name of the selected component appears in the text box next to **Browse**. You can click **Clear** to clear the name and browse again for the replacement model.
5. Click **Apply**. The name of the replacement model appears in the Component Replace column in the Model

- Tree next to the name of the component being replaced.
6. Modify the list of components to be replaced:
    - Use the SELECT MDL and GET SELECT menus to select additional components to replace.
    - Use the REPLACE COMP menu to add or remove items from the list displayed in the Component Replace column in the Model Tree.
  7. Choose **Done** from the REPLACE COMP menu. The system replaces the original component. The Component Placement dialog box opens.
  8. Constrain the replacement model, and click **OK**.  
The replacement model appears on the assembly displayed in the graphics window. The system swaps the new component into the assembly in the same place geometrically, and into the Model Tree.
  9. You can continue to replace additional components.
  10. Choose **Done/Return** from the ADV COMP UTL menu and **Done/Return** from the COMPONENT menu.

## About Copying Components

To create multiple independent instances of components in an assembly, you can use the **Copy** command in the ADV COMP UTL menu. The **Copy** command uses a coordinate system as the basis for translating or rotating the components. You must modify, replace, or delete the copied components one at a time.

**Note:** When you modify a component, all dimensions used to place the component appear for modification.

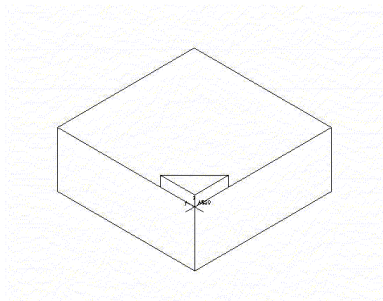
Each component has its own coordinate system, but the system assembles copies of the component into an assembly coordinate system. For example, when copying a component rotationally, Pro/ENGINEER does not rotate each copy around the coordinate system of the copied member; it rotates the copy around the coordinate system of the assembly. Generally, members use the same dimensioning references as the leader. Rotate the component (leader and all members) around the coordinate system of the assembly first; then translate (move) it.

## To Copy a Component

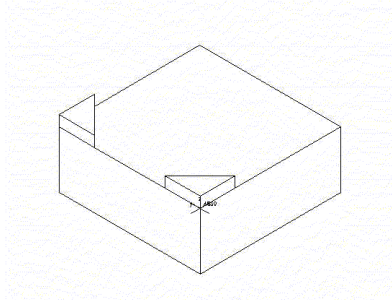
1. After assembling the component, choose **Component > Adv Utils > Copy**.
2. Create or select an assembly coordinate system.
3. Select the component(s) to copy. The EXIT menu and TRANS DIR menu appear.
4. Specify the moves using the following options from the EXIT menu to create the additional components. As in patterns, specify any number of incremental changes for a move in different directions. You can use any number of instructions per direction, but you can specify a maximum of three directions.
  - **Translate**—Patterns the component in the direction of the specified axis.
  - **Rotate**—Patterns the component about the specified axis.If you choose **Rotate**, the ROTATE DIR menu appears.  
Choose **X Axis**, **Y Axis**, or **Z Axis** from the TRANS DIR menu or the ROTATE DIR menu.
5. As you finish each set of moves, choose **Done Move** from the EXIT menu.
6. Specify the number of instances to create along this direction and repeat steps 4 and 5 to define the next copy direction. Continue this process until you have placed all copies.
7. Choose **Done** from the EXIT menu to execute all the moves.

## Example: Copying a Component

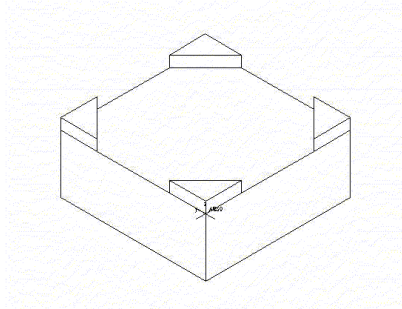
Assemble the component in any manner. Then add an assembly coordinate system.



The following figure illustrates the first move:



The following figure illustrates the completed copy:



## About Merging or Cutting Out Components

Using the **Merge** and **Cut Out** commands in the ADV COMP UTL menu, you can add or subtract the *material* of one set of parts to or from another set of parts, after you have placed them together in an assembly.

The **Merge** command adds the material from every one of the second set of parts selected into every one of the first set of parts. Depending on the additional options available, you can copy the features and relations of the second set of parts into each of the first set of parts or reference them by the first set of parts. This procedure creates the feature called *merge*.

The **Cut Out** command subtracts the material of every one of the second set of parts selected from every one of the first set of parts. In the same way as when using **Merge**, depending on the additional options selected, you can copy the features and relations of the second set of parts into the first set of parts or reference them by the first set of parts. This procedure creates the feature called *cutout*.

Use **Query Sel** to pick a merged part during assembly modification. The first set of parts in the assembly contains those parts you are going to modify, add material to, or remove material from. This set also contains the first parts that you selected when creating the merged or cut out part. The second set of parts contains the geometry that you are going to add to or remove from the first set of parts. This set also contains the second parts that you selected during this process.

When the parts you are merging have different accuracies, a message appears indicating the accuracy of the new part, up to a maximum of six decimal places. To undo or remove merges or cut outs, delete the merge/cutout features in the first set of parts.

**Note:** If you merge parts using the **Merge** or **Cut Out** options, mirroring geometry, or adding assembly features, the system does not show the geometric tolerances attached to the merged model dimensions in the drawing.

## Restrictions on the Merge and Cut Out Process

The parts used in a merge or cutout process must meet the following restrictions.

The first part cannot be one of the following:

- An assembly datum plane
- A subassembly
- An instance of a generic part (copy the part instance to another name and use the copy)
- A sheet metal part

The second part cannot be one of the following:

- An assembly datum plane
- A subassembly
- The same part as part 1 (itself)
- Part of an assembly pattern

In addition to the above restrictions, the following rules apply:

- If the second part references assembly members other than part 1, the assembly must be in session in order for the system to update the feature placement, and the second part must be attached to the assembly.
- The referenced part should only reference the part being cut out. If it references other components in the assembly, the system creates a reference assembly. If you then attempt to retrieve the merged or cut out part without the reference assembly in session, it warns you that it cannot update feature placement.
- If you place the second part into the assembly using the **Coord Sys** placement constraint, all the references of the assembly coordinate system used for placement must belong to the first part.
- If you assemble a component that you have cut out to assembly datum planes, the system creates it without offset dimensions for component placement.
- You must assemble the parts explicitly; you cannot use a part if you assembled it using packaging options.
- After you make a cut out, delete the second part or suppress it from the assembly to immediately view the new part.
- If the first feature of a merged part is a user-defined feature, or the result of using **Copy** in the **Adv Comp Util** menu, the system automatically incorporates all associated features into the merged part.
- Features that you have merged using **Copy** cannot reference features that are outside of the merge group. If you try to merge a component with a part that has an imported feature, the **Copy** command is dimmed in the **OPTIONS** menu.
- If you apply shrinkage by dimension to the reference model used in a cut out procedure, the model containing the cut out feature does not reflect the shrinkage dimensions. The system uses the original dimensions (before it applies shrinkage).

## To Merge or Cut Out Two Parts in an Assembly

The merge and cut out processes are exactly the same. To do either process, you use the same method.

1. Choose **Component > Adv Utils > Merge** or **Cut Out**.
2. Select the first set of parts. Choose **Done Sel**.
3. Select the second set of parts. Choose **Done Sel**.

4. For each combination of parts from the first set and second set, the **OPTIONS** menu appears. Choose the options for that specific combination of parts.
    - **Reference**—References the second part to obtain its information. When the referenced part changes, the merged or cut out part changes.
    - **Copy**—Copies all the features and relations of the second part into the first.
    - **No Datums**—Does not include datums of the second part in the merged part (available for merge by reference only).
    - **Copy Datums**—Includes the datums belonging to the second part in the merged part.
  5. Choose **Done** from the **OPTIONS** menu, and **Done/Return** from the **ADV COMP UTL** menu.
- Note:** After you assemble components before performing a cutout procedure, hidden line removal may not function as you expect. If the assembly components physically intersect, one or both of them may not appear correctly. If this occurs, you have not placed them correctly.

## Merging or Cutting Out by Reference or by Copy

### Merge or Cut Out by Reference or by Copy

It is possible to create a merge or cutout feature from the **Component/Adv Util/Merge or CutOut** menu picks. Here, you must specify the component to add the merge feature to, and then select the component to merge into the first component. A Merge or Cutout by Reference creates a feature that is similar to the Merge and Cutout feature from the Shared Data group.

When you merge or cut out using the **Reference** option, the system places the second member into the merged part by referencing the original second part used in the assembly.

When you merge or cut out using the **Reference** option, the system places the second member into the merged part by referencing the original second part used in the assembly.

The following rules apply for using the **Reference** option:

- The system reflects all eventual changes to the referenced part in the merged part.
- You *cannot* directly modify the features of the second member of the merged part, with the exception of the placement dimensions. However, you can modify them using **Query Sel**; the system then automatically reflects all the modifications in the reference part.
- You can redefine the second member of the merged part so that you can replace it with another instance of the family table.
- If you delete or suppress the geometry in a referenced part that places the reference member in the merged part, during the next regeneration of the merged part, the system places the member using the last successful placement, and it warns you that it cannot update feature placement. When you resume the suppressed geometry, it restores parametric placement.
- If you have deleted or renamed a referenced part, or it does not exist in the current directory, the system retrieves the merged part with a warning "Reference part *partname* is missing," and it does not regenerate it. (If you made a regeneration attempt and it failed, choose **Quick Fix** from the **FIX MODEL** menu; then choose **Suppress** from the **QUICK FIX** menu to suppress the corresponding merge member.) To correct the situation, locate and restore the referenced part.
- If a merged part *is active in the current session*, you can rename the referenced parts using the **Rename** command in the **File** menu, and the merged part updates its references. You cannot delete the referenced parts from memory.
- The system copies cosmetic feature geometry in the second part to the first part.
- Surface features that are outside a model show up in the model when a cut out is performed on components in an assembly.
- When saving a merged part, or defining a user-defined feature, the system automatically saves the parts referenced by the merged part.

## Merge or Cut Out by Copy

When you merge or cut out using the **Copy** command in the **OPTIONS** menu, you can modify all features of the merged part as in any other part; however, two rules apply:

- Except for placement dimensions, you cannot dimension features within a merge member to features outside that member. Therefore, when you modify the scheme of a merged part, only those features within one merge member are displayed in the window.
- The system adds features of the second merge member to the end of the feature list of the first part. As a result, when you set a feature of the second merge member as read-only, all the features of the first merge member also become read-only.

## Including Datums

When you merge or cut out two parts in an assembly, you must specify **Reference** or **Copy** from the **OPTIONS** menu for each combination of parts from the first set and second set.

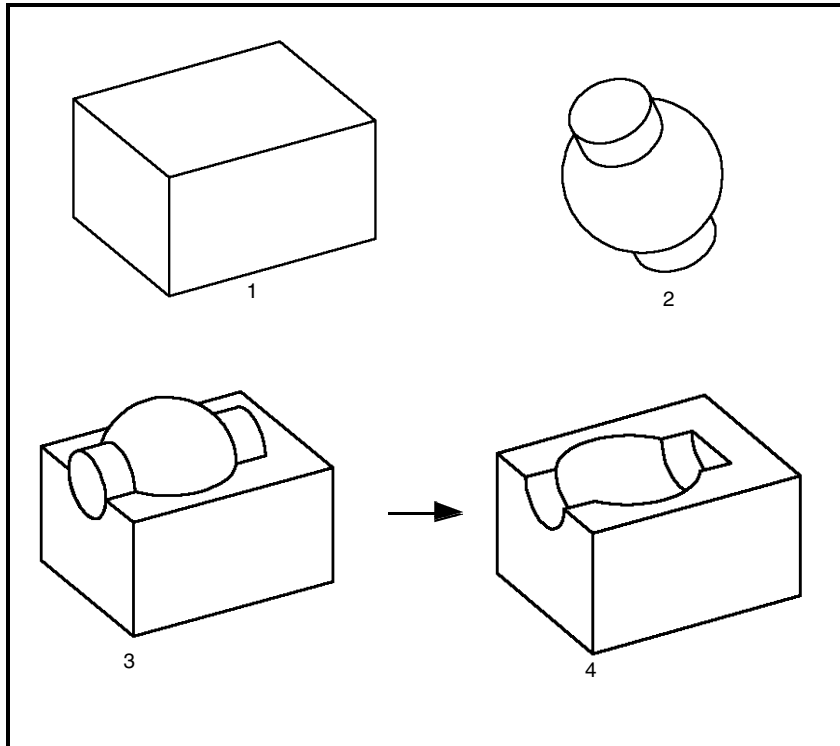
When you merge by reference, the **No Datums** and **Copy Datums** commands are available in the **OPTIONS** menu. You cannot modify included datums. When you cut out by reference, the system excludes datums. When you choose **No Datums**, the system excludes only datum planes from the merge; it merges datum axes, datum points, and coordinate systems.

When you merge or cut out by copy, the system always copies datums, and they belong to the merge or cut out feature.

## Example: Merged and Cut Out Parts

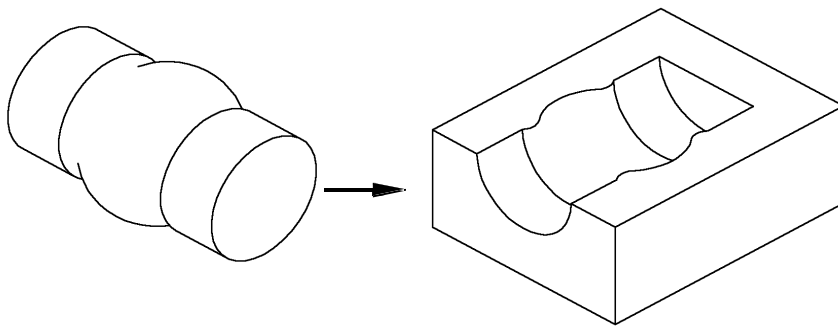
The following figure illustrates creating a cut out part.





- 1 First part
- 2 Second part
- 3 The parts explicitly assembled
- 4 Cut out part after deleting/suppressing second part

The following figure illustrates modifying a part referenced by a merged or cut out part. Changes to the second part—when it is referenced by the merged or cut out part—are automatically reflected in the merged part.

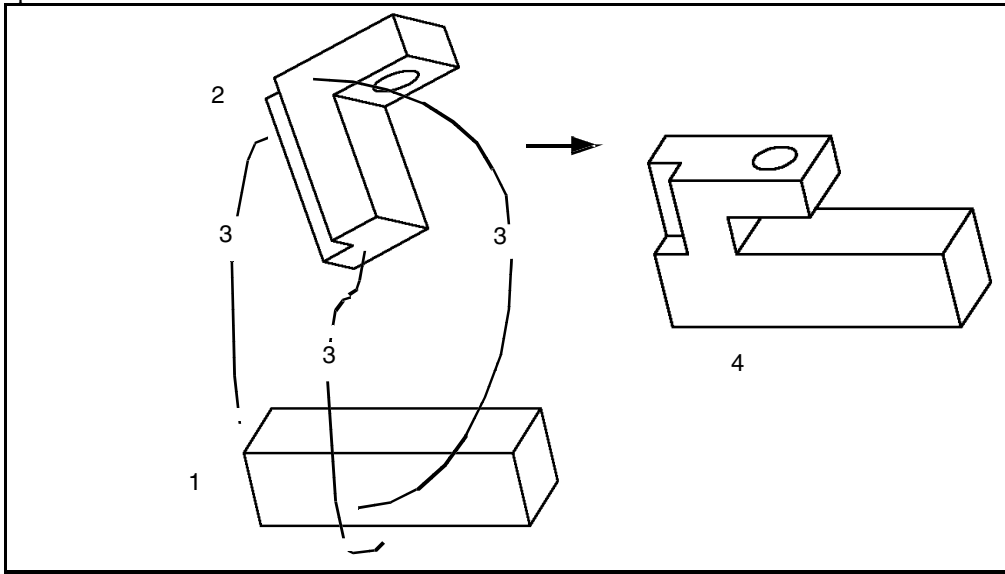


The following figure shows a merged part and illustrates merging by reference.

To create the example shown in the following figure, you assemble Part A and then merge it with Part B using the **Copy** option. The new part A now has four features. Features that belonged to Part B are the copied features and form a merged member. This merged member is like a subpart. The features of Part B regenerate first and then intersect with the features of Part A.

Note that the **Thru All** hole and cut that was copied does not extend through the original surfaces of Part A. Whenever a solid portion of one part intersects with a void (for example, a hole or cut) portion of another part, Pro/ENGINEER fills these areas with solid. However, if you now create a **Thru All** hole in the new Part A, it penetrates all features regardless of origin. You can merge merged parts with other parts, creating multiple

subpart relationships. For example, you could merge the new Part A with Part C, using the **Copy** or **Reference** option.



- 1 First part (A)
- 2 Second part (B)
- 3 Align
- 4 Merged part

## To Replace a Component by a Shrinkwrap Model

This method of replacement allows you to replace a master model with a shrinkwrap and vice versa while still maintaining all valid references. Because the system maintains references, it knows automatically how to place the new component. You can also replace one shrinkwrap with another and maintain references.

1. Retrieve and assembly.
2. Click **Component > Adv Utils > Replace**; or select one component from the Model Tree, and choose **Replace** from the pop-up menu.  
The **SELECT MDL** menu and the **Replace Component** dialog box open. An empty Component Replace column is displayed in the Model Tree.
3. Select the component to be replaced. This can be a shrinkwrap or a master model.
4. Click **By Shrinkwrap** in the **Replace Component** dialog box.
5. Click **Browse** to open the **File Open** dialog box.
6. Select the replacement model that is either a shrinkwrap part (a part with an external shrinkwrap feature of the model you are replacing), or a master model of the shrinkwrap part you are replacing.
7. Click **Apply**. The name of the replacement model appears in the Model Tree next to the name of the component it is going to replace.
8. Choose Done from the **REPLACE COMP** menu. The system swaps the new component into the assembly and into the Model Tree.

## Using the Mirror Subassembly Dialog Box

When you mirror an existing subassembly, this dialog box provides a convenient way to rename the mirrored components. The source subassembly is shown in a tree-like structure so you can see its hierarchy of components.

By default each component is selected to be mirrored and given a default new name. For each component, you

can choose to give it a new name, to exclude it from the mirroring operation, or to reuse the component in the source subassembly.

In the Rule section of this dialog box, you can establish a rule for a renaming convention. The default renaming convention is the old name appended with a default suffix. You can change the suffix and apply it to selected components. You can also create a template for other renaming conventions.

## To Place an Assembly User-Defined Feature

When placing Assembly UDFs, you must select all prompted references as with any other UDF. If the UDF contains an Assoc Model, the element Assoc Mdl Name is added to the Group ODUI and you can rename any Assoc Models in the UDF.

1. Click **Assembly > Feature > Create > User Defined**. The file browser opens in the Group Directory.
2. Select the UDF name and click **Open**. The system opens the PLACE OPTS menu with the following options:
  - **Independent**—Copy all the required UDF values into the part to create a group that is independent of any changes to the UDF.
  - **UDF Driven**—The group remains driven by the UDF. The dimensions of a UDF-driven group update either automatically whenever the model is retrieved, or when you choose **Update** from the Group menu.
3. Specify the placement scale by choosing one of the options in the SCALE menu, followed by **Done**.
4. Select the display option for invariable dimensions by choosing an option from the DISP OPTION menu, followed by **Done**.
5. Place the UDF by selecting placement references. As each placement prompt appears, select an action from the SEL REF menu.
6. In the GP MODEL NAMES dialog box, select new names or accept the default names for any Assoc Models in the UDF.
7. RegenAction regenerates all of the copied models.
8. Click **OK** to create the group.

## About Defining Interface Constraints

An interface is a partially defined set of constraints that defines how a component should be assembled to another component or into an assembly. As an illustration, suppose the user knows that certain surfaces of component A will be used for creating mate/align constraints while assembling it, he can create an interface with the mate/align constraints and relate these surfaces as component references to the constraints. This constraint set is called an interface. A user can create and store multiple interfaces in a component before the component is assembled. A component may contain many interfaces, each defining a different way to assemble the component. Saving this information with the component can increase the ease of automatic assembly and design integrity.

When you define an interface, the constraints available for use are the same as the existing component placement set. The existing topology selection filtering that exists in component placement occurs here. For example, if you choose an Insert constraint, a cylindrical surface must be selected as the component reference.

After an interface is defined for a component, the interface is available to use during future component placements. When assembling a component with stored interfaces, the Select Interface dialog box appears to allow the selection of an existing interface. If an interface is selected, the component constraints are automatically populated in the Constraints list. The user only needs to select the matching assembly constraints. Once in the Component Placement dialog box, clicking the **Interface** button opens the Select Interface dialog box which contains the list of interfaces defined for this component.

In Pro/ENGINEER 2001, only component interfaces can be stored. If interfaces are stored with an assembly, they are only be accessible when that assembly is being assembled to another assembly.

# To Define an Interface for a Component Using Set Up

1. On the **ASSEMBLY** or **PART** menu, click **Set Up** to open the **ASSEM SETUP** or **PART SETUP** menu.
2. Click **Comp Interface**. The **Interface Definition** dialog box opens.

The default name of the first interface is listed in the Interface Names section. This name can be edited by clicking on it.

If an Interface has already been defined, the constraints of that interface will be listed in the constraint Type list. Picking on a constraint in that list will highlight the corresponding reference in the graphics window. The **Add** and **Remove** buttons add new interface names to the list or remove existing ones.
3. In the **Type** list the constraint type **Mate** appears as the default. To select a different constraint type, double-click on the word **Mate** and the list of other constraint types appears. The available constraint types are:
  - **Mate**—References must be of the same type. Two surfaces will touch each other, facing each other.
  - **Align**—2 planes are coplanar, 2 axes coaxial, or 2 points coplanar.
  - **Insert**—Insert one revolved surface inside another revolved surface.
  - **Coord Sys**—The component will be placed in an assembly with the component's coordinate system aligned with the assembly's coordinate system.
  - **Tangent**—Controls the contact of two surfaces at their tangency.
  - **Pnt On Line**—Controls the contact of an edge, axis or datum curve with a point.
  - **Pnt On Srf**—Controls the contact of a surface with a point.
  - **Edge on Srf**—Controls the contact of a surface with a planar edge.
4. Select the component reference.
5. Add additional constraints and references until the interface is sufficiently defined.
6. Click **OK** to store the interface with the part.

## Creating Merge and Cut Outs from Shared Data Menu

You can perform the following functions when creating a Merge or Cut Out feature in a part:

- Merge
- Merge from Other Model (ExtMerge)
- Cutout
- Cutout from other Model (ExtCutOut)

The Merge and Cutout features create geometry using placement in an existing assembly. Merge for Other Model and Cutout from Other Model allows you to specify the location of the Merge or cutout reference using coordinate systems or use the Default placement. It is possible to externalize a Merge feature, which removes the dependence to the assembly. When externalizing a Merge or Cutout feature, location must be specified using coordinate systems, default placement or assembly relative location may be frozen. You may also specify whether the merge or cutout geometry will be dependent on the reference part. If a merge feature is independent, you cannot externalize the feature.

You also have the option of copying datums if the merge feature is dependent. If the datums are copied, refit datum options appear in the **Merge** dialog box. If the feature is independent, no datums are copied. Datums may be refitted with existing geometry only.

## To Define an Interface During Component Placement

1. Assemble a component using **Component Placement** dialog box. Once you define all the constraints in the constraints list, select the Interface icon.
2. The **Select Interface** dialog box opens. If the check box for **Add New Interface** is checked, the constraints from the Component Placement dialog box will be saved to the new interface name in the text box below the checkbox icon. Click the default name to edit the new interface name.
3. Click **OK** to save the constraint to the new name.

## About Modifying an Assembly at Top, Subassembly, or Part Level

When you make a modification to a component in Assembly mode, the instance is updated automatically in Part and Drawing modes.

To modify an assembly or any part in an assembly, choose **Modify** from the ASSEMBLY menu to display the ASSEM MOD menu with the following commands:

- **Mod Part**—Allows you to modify parts in the assembly and to add, modify, or delete features.
- **Mod Skel**—Allows you to modify skeleton models.
- **Mod Subasm**—Allows you to modify features and offset dimensions of any subassembly in the top-level assembly and to assemble components into the subassembly.
- **Mod Assem**—Allows you to modify only the top-level assembly dimensions.
- **Mod Dim**—Allows you to modify any dimension in the assembly.
- **Mod Expld**—Allows you to modify explosion distance dimensions.

You can also select a component in the Model Tree and then choose **Modify** from the Model Tree pop-up menu to access the ASSEM MOD menu.

## To Modify an Assembly

Using the **Mod Assem** command from the ASSEM MOD menu, you can modify assembly features and offset dimensions directly in the window in which the assembly currently displays.

1. Choose ASSEMBLY > **Modify**, or choose **Modify** from the Model Tree pop-up menu. The ASSEM MOD menu appears.
2. Choose **Mod Assem**. The MODIFY ASSY menu appears.
3. Choose one of the following:
  - **Move**—Moves members in the assembly using reference coordinate systems.
  - **Modify Dim**—Modifies assembly dimensions and features using the MODIFY menu.
  - **Regenerate**—Updates the assembly and selected parts. Displays the PRT TO REGEN menu to select individual parts to regenerate or to instruct the system to regenerate all necessary parts automatically.

## To Modify a Subassembly

Using the SUBMODEL menu, you can modify subassembly features and offset dimensions directly in the window in which the subassembly currently displays. The **Copy From** command is not available.

1. Choose ASSEMBLY > **Modify**, or choose **Modify** from the Model Tree pop-up menu. The ASSEM MOD menu appears.
2. Choose **Mod Subasm**; then select the subassembly to modify. The SUBMODEL menu appears.

3. Choose any of the following commands:
  - **Component**—Manipulates assembly components using the COMPONENT menu. You can use the **Copy**, **Merge**, and **Cut Out** commands only in the top-level assembly.
  - **Feature**—Manipulates assembly features using the ASSY FEAT menu.
  - **Modify**—Modifies assembly or assembly component dimensions and features using the MODIFY menu.
  - **Design Mgr**—Accesses tools to manage assembly design. You can control creation of external references and investigate references and dependencies. You can use the **Envelope** and **Zone** commands only in the top-level assembly.
  - **Regenerate**—Updates modified part and assembly dimensions. Displays the PRT TO REGEN menu to select individual parts to regenerate or to instruct the system to regenerate all necessary parts automatically.
  - **Relations**—Displays the MODEL REL and RELATIONS menus to edit parametric labels and add or edit constraint equations.
  - **Family Tab**—Displays the Family Tree dialog box to edit the assembly family table or create assembly instances.
  - **Show Model**—Displays the model you are changing.
  - **Set Up**—Sets up assembly mass properties using the ASSEM SETUP menu.
  - **Program**—Displays the PROGRAM menu to use Pro/PROGRAM capabilities.

## To Modify Dimensions of a Part in an Assembly

Using the **Mod Dim** command from the ASSEM MOD menu, you can display and change dimensions of any or all parts in an assembly.

Note that you can display the dimensions in only one window at a time; that is, you cannot display the dimensions of a part feature in a part window and in an assembly window at the same time.

1. Choose ASSEMBLY > **Modify**, or choose **Modify** from the Model Tree pop-up menu. The ASSEM MOD menu appears.
2. Choose **Mod Dim**; then select the feature to modify, select the dimension, and enter a new value.

**Note:** You must use the **Regenerate** command to update the modified parts after you make dimensional modifications. You can select individual parts to regenerate; however, the parts chosen this way will be regenerated in the order in which they were chosen.

## Tip: Regenerate After Changing a Dimension

It is essential to regenerate whenever you change a dimension.

## To Display Default Dimensional Tolerances

To turn on dimension tolerance display, use one of the following methods:

- Select **Dimension Tolerances** in the **View** menu Model Display dialog box
- Select **Dimension Tolerances** in the **Utilities** menu Environment dialog box
- Set the configuration file option `tol_display` to yes.

## To Modify Default Dimensional Tolerances

When dimension tolerance display is enabled, you can display and modify default dimensional tolerances for an

assembly or a component in Assembly mode.

To modify the tolerance values for an assembly:

- Select a model from the Model Tree, and choose **Modify** from the pop-up menu, or retrieve a model, and choose **ASSEMBLY > Modify > Mod Dim**.  
The tolerance values displayed are for the top-level assembly, not for the selected component.
- Select a tolerance from the display of tolerance values in the lower right corner of the graphics window, and enter a new value.

To modify the tolerance values for a component:

- Choose **Mod Part**, **Mod Skel**, or **Mod Subasm**, and then select a part, skeleton model, or subassembly.  
The tolerance values displayed are for the selected component.
- Select a tolerance from the display of tolerance values in the lower right corner of the graphics window, and enter a new value.  
When you exit from this menu, or from other operations such as **Redefine**, **Suppress**, or **Resume**, the tolerance values switch back to those of the assembly level—your modified values exist, but the values displayed are those of the assembly level.

## Assembly and Component Default Dimensional Tolerances

You can display and modify default dimensional tolerances for an assembly or for a component in Assembly mode. If dimension tolerance display is turned on, the default dimensional tolerances are displayed on screen, below the model, in the lower area of the graphics window, as they are in Part mode. All configuration file options used to specify the default values for tolerances are interpreted in both Part mode and Assembly mode. The on-screen display always accurately presents the default dimensional tolerances for the object being modified, whether that object is a part, an assembly, or a part being modified in the context of an assembly. You cannot select dimensions outside the part or assembly that you are modifying. For example, if you choose **Modify > Mod Part**, you can modify only the dimensions on the selected part.

## To Set Relative Accuracy for a Model in Assembly Mode

1. Choose **ASSEMBLY > Set Up**. The **ASSEM SETUP** menu appears.
  2. Choose **Accuracy**.  
When the configuration file option `enable_absolute_accuracy` is set to `no`, the system displays the current relative accuracy of the active model and prompts you to change it.  
When the configuration file option `enable_absolute_accuracy` is set to `yes`, the **ACCURACY** menu appears. Choose **Relative**.
  3. The system displays the current relative accuracy of the active model and prompts you to change it.
  4. Enter a value for relative accuracy. The system informs you that changing accuracy causes full regeneration and asks if you want to continue.
  5. Click **Yes** (the default). If you click **No**, the modification does not take effect.
- You can accept the current value, or press **ESC** to return to the **ACCURACY** menu without changing the value. You can choose **Quit** to return to the **ASSEM SETUP** menu whenever the system prompt is inactive.

# To Set Absolute Accuracy for a Model in Assembly Mode

**Note:** The configuration file option `enable_absolute_accuracy` must be set to `yes` to enable modification of absolute accuracy in Assembly mode.

1. Choose **ASSEMBLY > Set Up**. The **ASSEM SETUP** menu appears.
2. Choose **Accuracy**. The **ACCURACY** menu appears.  
The first time you access the **ACCURACY** menu for a model, **Relative** is active by default. Thereafter, the accuracy type you used last is active when you access the **ACCURACY** menu. If **Relative** is active, press **ESC**.
3. Choose **Absolute**. The **ABS ACCURACY** menu appears. Do one of the following:  
Set absolute accuracy by specifying a value:
  - Choose **Enter Value**.  
The system displays the current absolute accuracy value and default units [nnnn units] and prompts you to enter a value. If absolute accuracy has not been defined, the system displays the current relative accuracy. If an absolute accuracy value has not been previously defined, the assembly unit is highlighted in the message prompt.
  - Enter a value for absolute assembly accuracy in default units. The system informs you that changing accuracy causes full regeneration and asks if you want to continue.
  - Click **Yes** (the default). If you click **No**, the modification does not take effect.

Set absolute accuracy by assigning a value:

- Choose **Select Model**. The File Open dialog box opens.
- Select a source model. The system displays the current accuracy value of the specified model and prompts you to accept it. If you click **Yes**, the system informs you that changing accuracy causes full regeneration and asks if you want to continue.
- Click **Yes** (the default). If you click **No**, the modification does not take effect.

You can accept the current value, or press **ESC** to return to the **ACCURACY** menu without changing the value. You can choose **Quit** to return to the **ASSEM SETUP** menu whenever the system prompt is inactive.

**Note:** Selecting a part or assembly as a source for the absolute accuracy setting does not create dependency between the two models.

## Modifying Accuracy Settings in Assembly Mode

Pro/ENGINEER allows you to modify accuracy settings in Assembly mode, as well as in Part, and Manufacturing modes.

When geometry is copied from one part to another with different absolute accuracy, the source geometry can be invalid for the destination part. Online Help documentation for Part mode provides detailed information about modifying part accuracy settings.

An assembly can have an accuracy setting that is incompatible with that of a part. For example, an assembly hole or cut feature may not display because the accuracy settings of intersected parts are too high. When assembly accuracy and part accuracy differ, you can reset accuracy settings easily at the assembly level.

The configuration file option `enable_absolute_accuracy` must be set to `yes` to enable modification of absolute accuracy in Assembly mode.

When the configuration file option `enable_absolute_accuracy` is set to `no`, you can modify only relative accuracy. The system displays the current relative accuracy of the active model and prompts you to change it. The default setting 0.0012 is the initial setting.

When the configuration file option `enable_absolute_accuracy` is set to `yes`, the **ACCURACY** menu appears, containing the **Relative** and **Absolute** commands.



## To Modify a Part Feature

Using the **Mod Part** command in the ASSEM MOD menu, you can create, delete, suppress, and modify part features in Assembly mode. The **Copy From** command is not available.

**Note:** When you create part features at the assembly level, be careful not to create unwanted parent/child relationships between the part and the assembly.

1. Choose ASSEMBLY > **Modify**, or choose **Modify** from the Model Tree pop-up menu. The ASSEM MOD menu appears.
2. Choose **Mod Part**; then choose a part to modify. The MODIFY PART menu appears.
3. Choose one of the following:
  - **Feature**—Adds a feature to the specified part in assembly mode (displays the PART FEAT menu).
  - **Modify Dim**—Modifies the dimensions of the specified part (displays the MODIFY menu).
  - **Regenerate**—Regenerates the specified part.
  - **Ref Control**—Controls creation of external references (displays the Reference Control dialog box).
  - **Global Ref Viewer**—Displays references and dependencies (opens the Global Reference Viewer dialog box).

## Creating or Deleting Part Features

Feature creation and deletion is the same as in Part mode except for a few rules specific to Assembly mode:

- You can work on only one part at a time. You select this part before feature modification begins, and it is the active part. You can select an active part by choosing ASSEMBLY > **Modify** > **Mod Part**. All other parts in the assembly remain inactive until you complete feature modification.
- Features created cannot traverse parts. Feature creation only affects the active part.
- Datum planes and coordinate systems created using the MODIFY PART menu pertain to the active part. They are not assembly datums or coordinate systems.

## To Modify a Skeleton Model

Using the SKEL OPER menu, you can modify an existing skeleton model in an assembly or subassembly. You can work on the skeleton model in the context of the assembly or in a separate subwindow, but only the features that belong to the skeleton actually appear in the subwindow.

**Note:** The commands in the ADV COMP UTL menu are not available for skeleton models. You cannot copy a skeleton model, create a group or user-defined feature with a skeleton as one of its members, or perform merge and cut operations.

1. Choose ASSEMBLY > **Modify**, or choose **Modify** from the Model Tree pop-up menu. The ASSEM MOD menu appears.
2. Choose **Mod Skel**; then choose a skeleton model to modify. The SKEL OPER menu appears.
3. Choose one of the following commands, and proceed as you would for any other component:
  - **Feature**—Creates, deletes, redefines, and manipulates skeleton model features. Displays the FEAT menu.
  - **Modify**—Modifies dimensions, dimension formats, and geometric tolerances. Displays the MODIFY menu.
  - **Regenerate**—Updates skeleton models as specified.
  - **Relations**—Displays the MODEL REL and RELATIONS menus to edit parametric labels and add or edit constraint equations.
  - **Set Up**—Sets up units, parameters, notes, and reference dimensions. Displays the SKEL SETUP menu with the following commands:
    - **Material**—Creates or modifies material data (displays the MATRL MGT menu).
    - **Units**—Specifies units to use (opens the Units Manager dialog box).

- **Name**—Specifies names for a feature, surface finish, primary value of a geometric tolerance, axis, curve, or other item (displays the NAME SETUP menu).
- **Parameters**—Specifies parameters (displays the OBJ TYPES and MODEL PARAMS menus).
- **Notes**—Adds, removes, or modifies notes associated with the skeleton model (displays the MDL NOTES menu).
- **Ref Dim**—Creates reference dimensions (displays the REFDIM, REF TYPE, and VIEW NAMES menus).
- **Geom Tol**—Specifies the geometric tolerances to a surface or feature.
- **Declare**—Declares a datum plane or axis to be related to a global datum plane or axis (displays the DECLARE menu).
- **Tol Setup**—Specifies tolerances standard.
- **Ref Control**—Controls creation of external references. Opens the Reference Control dialog box.
- **Program**—Accesses Pro/PROGRAM.
- **X-section**—Creates a cross-sectional view. Displays the CROSS SEC and XSEC NAMES menus.

## Skeleton Models and Assembly Features

Because the skeleton model geometry is not regular geometry of the assembly, it is not affected by assembly-level features. Assembly features such as cuts and holes do not intersect the skeleton model geometry.

If you want to intersect the skeleton model with a cut, choose **Modify > Mod Skel**, and select the skeleton model; then create a cut in the skeleton model.

## About Assembly Features

**Note:** Click *See Also* for detailed information about subtractive features.

You can create and modify assembly features, and you can use assembly features in Part mode.

Assembly features are displayed when you retrieve a model.

You can create assembly datum features—assembly datum planes, axes and points, curves, and coordinate systems—using the basic functionality of Pro/ENGINEER. These entities differ from those created in Part mode in the sense that they belong to the assembly and not to a certain part. The process of their creation is very much like that for Part mode, but there are some differences and restrictions.

Assembly datums are particularly useful for the following:

- Creating assembly cross-sectional views
- Creating zones of the assembly
- Placing components at an offset or angle from other components
- Creating a reference datum plane whose constraints go across two components

Pro/ENGINEER labels them as ADTM# (for example, ADTM1, ADTM2, ADTM3), ACSYS#, APNT#, and so on. If you write relations using an assembly datum plane, the system stores them with the assembly.

You can create assembly datum features at any time. If you have not yet placed the base component in the assembly, you can create three orthogonal datum planes using the ASSY FEAT menu without creating a corresponding part in Part mode.

You can create assembly features other than datum planes, such as the following:

- Subtractive features (hole, cut, slot)
- Pipe features
- Sketched cosmetic features
- User-defined features

The pipe feature is a three-dimensional centerline that represents the centerline of a pipe. Given the diameter of a pipe (and, for a hollow pipe, the wall thickness), a pipe connects selected datum points either with a combination of straight lines and arcs of specified bend radius, or a spline.

You can construct pipes in Assembly mode as either a part feature or an assembly feature. When you create a pipe as a part feature in Assembly mode, you can use datum points on other parts. A pipe can also be an Assembly feature although it will have no geometry, that is, solid assembly pipe feature geometry does not display on the screen. Only the pipe centerline displays.

If you have a license for Advanced Pro/ASSEMBLY Extension, you can create user-defined features; without a license, you can only place user-defined features.

## To Create an Assembly Feature

**Note:** Click *See Also* for detailed information about subtractive features.

1. Choose **ASSEMBLY > Feature > Create**, or select the assembly and choose **Feature** from the pop-up menu in the Model Tree. The FEAT CLASS menu appears.
2. Choose a command from the FEAT CLASS menu, and then define one of the following features, in the same way that you create a feature in Part mode:
  - Pipe and cosmetic features
  - Datum features
    - When you create assembly datum curves at the intersection of surfaces, you must select assembly surfaces or features. You cannot reference part surfaces.
  - User-defined features that do not add material
3. Choose **Done** from the FEAT CLASS menu and **Done/Return** from the FEATURE menu.

## To Modify an Assembly Feature

You can modify assembly features in Assembly mode using **Modify Dim**. You can pattern, delete, suppress, and resume them as ordinary features.

1. Choose **ASSEMBLY > Modify**, or choose **Modify** from the Model Tree pop-up menu. The ASSEM MOD menu appears.
2. Choose **Mod Assem**. The MODIFY ASSY menu appears.
3. Choose **Modify Dim**. The MODIFY menu appears.

## To Copy an Assembly Feature

Choose **ASSEMBLY > Feature**. The **Copy** command in the ASSY FEAT menu displays the COPY FEATURE menu, enabling you to copy components and assembly features.

Online Help documentation for Part mode provides detailed information.

## Copying Assembly Features

Copying features in Assembly mode works in the same way as in Part mode, with the following exceptions:

- You cannot use the **All Feat** option.
- You cannot mirror assembly features.
- When you create a copy of an assembly feature, the system copies assembly relations with it, but they are not related to the feature.

# About Intersecting Components with Subtractive Assembly Features

You can create and modify assembly features, update old style assembly features, use assembly features in Part mode, and delete parts that assembly features intersect.

You can create subtractive assembly features (holes, cuts, slots). You can also modify subtractive assembly features by changing the parts to be intersected. For example, you can assemble new components and then intersect them with assembly features that already exist. You can also clear parts that an assembly feature already intersects.

## To Create a Subtractive Assembly Feature

1. Choose **ASSEMBLY > Feature > Create**, or select the assembly and choose **Feature** from the pop-up menu in the Model Tree. The **FEAT CLASS** menu appears.
2. Choose a command from the **FEAT CLASS** menu, and then define a subtractive feature—hole or cut—in the same way that you create a feature in Part mode.  
You can define subtractive assembly features as **Blind**, **Thru All**, or **UpTo** (**UpTo Pnt/Vtx**, **UpTo Curve**, **UpTo Surface**). These depth commands only represent an intended maximum of the depth of the feature. The actual display of a feature depends on which parts you have selected for it to intersect.  
**Note:** If you want an assembly feature to intersect all the parts eventually placed into an assembly, define it using the **Thru All** and **Both Sides** options; then add the parts to the intersection list that the assembly feature intersects.
3. After you finish defining a subtractive feature in Assembly mode, the **Intersected Comps** dialog box appears. You can use this dialog box to perform additional operations:
  - You can allow the system to make the assembly feature visible at the assembly top level, or you can specify the level at which the assembly feature is visible.
  - You can add components for the feature geometry to intersect.
  - You can allow the system to create new intersected component instances with system-defined names, or you can specify names for the intersected component instances. Specifying names controls the visibility of instances of the intersected components:
4. Choose **Done** from the **FEAT CLASS** menu and **Done/Return** from the **FEATURE** menu.

## To Add Components to Be Intersected

After you finish defining a subtractive feature in Assembly mode, the **Intersected Comps** menu appears. Or, choose **Feature > Intersect**, and then select an assembly feature. The **Intersected Comps** dialog box appears.

1. Click the selection arrow to open the **GET SELECT** menu. It allows you to manually select components from the Model Tree or the graphics window to add to the **ModelList**.
2. Click **AutoAdd** to select all the components with bounding boxes that intersect the assembly feature; then click **Accept** from the **CONFIRM** menu.

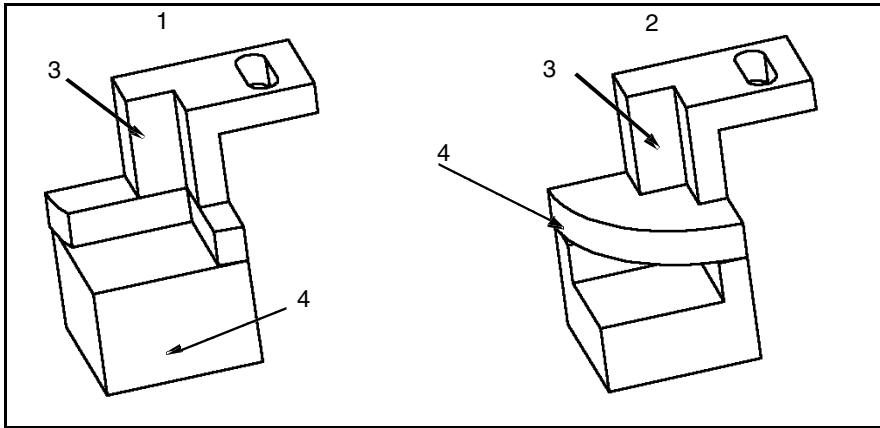
**Note:** If the system automatically selects a component that you do not want to include in the intersection, you can remove it using the **Remove** command in the **Intersected Comps** dialog box.

## Tip: Modify a Skeleton Model to Intersect with a Cut

If you want to intersect a skeleton model with a cut, choose **Modify > Mod Skel**, and select the skeleton model; then create a cut in the skeleton model.

## Example: Selecting Parts to Intersect

The following figure presents an example of selecting parts to intersect using the **Thru All** and **Blind** options.





- 1 Thru All
- 2 Blind
- 3 Cut
- 4 Part not selected

## To Remove Intersected Components

After you finish defining a subtractive feature in Assembly mode, the **Intersected Comps** dialog box appears. Or, choose **Feature > Intersect**, and then select an assembly feature. The **Intersected Comps** dialog box appears.

Highlight a model in the Model List and click **Remove** to remove the model from the list.

Multiple models can be selected for removal from the list. Selecting a model name on the list also highlights the model on the graphics screen.

- Select All  selects all models in the list.
- Unselect All  clears the selection of all models in the list.

For assembly features created before Release 15.0, the old assembly feature intersection interface appears.

## To Add or Remove Parts Intersected Before Release 15.0

For assembly features created before Release 15.0, the old assembly feature intersection interface appears.

1. Choose **Add** or **Remove** from the INT PART menu.
2. Select the desired parts.
3. Select the level at which the feature should be visible.
4. Choose **Done Sel**.

Choose **View/Repaint** to display the changes.

## To Update to Remove Nonintersected Components

After you finish defining a subtractive feature in Assembly mode, the **Intersected Comps** dialog box appears. Or,


choose **Feature > Intersect**, and then select an assembly feature. The **Intersected Comps** dialog box appears.

Select the **Auto Update Intrsects** check box. This option performs a one time re-evaluation of intersecting components and removes any components that no longer intersect the feature.

For assembly features created before Release 15.0, the old assembly feature intersection interface appears.

## About Specifying Visibility Levels

Visibility levels are set in the **Intersected Comps** dialog box. The **Level** options set the visibility level of any part added to the ModelList.

- If you use the manual selection arrow  to add parts to the ModelList, each model is added with the current visibility level.
- If you use **AutoAdd**, all the automatically added models have the same visibility level.

If **Level** is set to **Top Level** or **Part Level**, the visibility level is shown as parts are added to the ModelList.

- **Part Level**—Makes the new feature visible wherever this version of the part is used, even outside the current assembly. This is similar to creating a feature using **Mod Part** in the **ASSEM MOD** menu.
- **Top Level**—Displays the feature only at the top-level assembly and creates instances of intersected parts. Does not create external references.

If **Level** is set to **Sel Level** for manual selection of parts, when you select a part the **SEL MEMBER** menu opens with all the available visibility levels for the part. You select the level of visibility that you want to apply to the part.

**Note:** When **Level** is set to **Sel Level**, the use of **AutoAdd** and **Auto Update** are not allowed.

- If the **Auto Update Intrsects** box is checked, you automatically update the intersections upon regeneration. You must regenerate to view results. Upon regeneration of the assembly with the assembly intersection with **Auto Update Intrsects** selected, any components added to the assembly before the assembly feature will automatically be intersected. Auto Update does not remove any previous components, and does not add any components that are assembled after the intersecting feature.
- If you specify new names for intersected component instances, the use of **AutoAdd** and **Auto Update** are not allowed.

**Note:** Click a component listed in the ModelName and VisLevel box to highlight the component in the graphics window and model tree.

## Intersection Visibility

The system stores the information used to create assembly feature intersection geometry at whichever level the assembly feature intersection appears. When selecting components for an assembly feature to intersect, you can specify the level at which the intersection should be visible and select all components automatically.

- If you display the intersection at the assembly level, the assembly feature is only visible when you are in that assembly (the part is not affected).
- If you display the intersection at the part or subassembly level, the assembly feature is visible at that level and all levels above it.

If you use the default options, the system stores all information in the top-level assembly. If you display the assembly feature in a lower-level assembly, the system creates an external reference to the top-level assembly (in the lower-level assembly). After modifying an assembly feature, you can automatically remove all components that the feature no longer intersects.

**Note:** Components selected for assembly intersections do not need to have the same type of length units.

## To Use System-Defined Names for Intersected Component Instances

After you finish defining a subtractive feature in Assembly mode, the **Intersected Comps** dialog box appears. Or, choose **Feature > Intersect**, and then select an assembly feature. The **Intersected Comps** dialog box appears.

1. Click to open the **New names** section of the dialog box. Default names are provided.
2. Click **OK**.

The system creates the new instances with system-defined names.

## About Intersected Components

When subtractive assembly-level features intersect a part in the assembly, the system automatically generates instances of the intersected components (parts and subassemblies).

If you use system-generated names (the default), created instances are invisible in family tables, the BOM, and Integrate Difference tables, and they do not function in the same way as regular family table instances.

Alternatively, you can name assembly feature component instances, and they are visible in family tables, the BOM, and Integrate Difference tables, and they function in the same way as any family table instances. If you do not enter a name for a subassembly model, the system uses the current name.

In either case, both the original and new instance models are in memory while the assembly feature is present.

You can use commands from the **Intersected Comps** dialog box to manipulate component instance name assignments:

- **Level**—The pull-down menu selects the assembly level at which the new names for the instances begin. The SEL MEMBER menu lists the levels.
- **New name**—In this section you can specify a new name for the instance to replace the current model name.
- **Level**—Highlights in magenta the model geometry in the Main window that corresponds to the current assembly level selected in the information window.
- **Info**—Displays the model information for the current assembly level.

## To Specify Names for Intersected Component Instances

After you finish defining a subtractive feature in Assembly mode, the INTR SCT OPER menu appears. Or, choose **Feature > Intersect**, and then select an assembly feature. The **Intersected Comps** dialog box appears.

1. Open the New name section of the dialog box.
2. Click the Selection arrow to open the **GET SELECT** menu.
3. Choose a model to add to the list of intersected components.  
The system creates the assembly feature intersection at the specified level.
4. Click on the VisLevel field of the model list to display available visibility levels for the model.
5. Select a new visibility level.
6. Click the ModelName field of the model list.
7. The model name is displayed as the OLDNAME in the New name list.
8. Enter a new name for the component, then click **OK**.

## To Show Intersected Components

After you finish defining a subtractive feature in Assembly mode, the **Intersected Comps** dialog box appears. Or, click **Feature > Intersect** and then select an assembly feature. The **Intersected Comps** dialog box appears.

Click a component listed in the ModelName and VisLevel box to highlight the component in the graphics window.

## To Display Information About Intersections

After you finish defining a subtractive feature in Assembly mode, the **Intersected Comps** dialog box appears. Or, click **Feature > Intersect**, and then select an assembly feature. The **Intersected Comps** dialog box appears.

To display information about the current status of assembly feature intersections, choose the **Info** command.

## To Change the Level of an Intersection

To display information about the current status of assembly feature intersections, choose the **Info** command.

1. Choose **Feature > Intersect**, then select the feature.  
The **Intersected Comps** dialog box opens.
2. Select a component from the model list.
3. Click on the VisLevel field next to the model name.
4. Click the arrow that appears in the VisLevel field to open a list of available visibility levels for this component.
5. Click **OK**.

## To Use Assembly Features in Part Mode

An assembly feature visible at the part level is a part feature visible in Part mode. When you suppress an assembly cut, it does not affect the parent assembly feature. When you delete an assembly cut in Part mode, the system removes the intersection from the parent assembly feature. If you select an assembly cut for modification (provided the parent assembly is in session), you can modify the dimensions of the parent feature in Part mode. In this case, regeneration only updates the current part. To update other parts intersected by this subtractive feature, regenerate them individually, or use the **Automatic** command to automatically regenerate in Assembly mode. Features in the intersected part should not directly or indirectly reference an assembly cut.

## To Retrieve and Reintersect Out-of-Date Assemblies

The **ReIntersect** command is available in the ASSY FEAT menu only if you have set the configuration file option `allow_redo_intersections` to yes.

1. Choose **ReIntersect** from the ASSY FEAT menu. The RE INTR menu displays the following options:
  - **All**—Updates intersections of all assembly features in the assembly.
  - **Sel Part**—Updates intersections of all assembly features intersecting selected parts.
  - **Info**—Displays an information window listing all assembly features in the assembly.
2. Choose **All** or **Sel Part**.
3. Select the assembly or a component of it.  
Before reintersecting, the system displays an information window. The message warns you that children not in session are going to fail regeneration and instructs you to retrieve all models into session that reference the geometry created by the assembly feature.
4. If all children are not in session, choose **Cancel** from the CONFIRMATION menu, retrieve the children that



are not in session, and return to Step 1. If all children are in session, choose **Confirm**, and all intersections are updated.

If remnants of assembly feature intersections exist, the system displays another information window. The message warns you that remnants exist, describes possible causes, and lists the particular assemblies that have intersection remnants. If you choose **Cancel**, the system does not remove remnants and some of the intersections may not be updated to the current style. Remove the remnants by choosing **Confirm**.

5. Once the intersections are updated, save all the models.

## Assemblies Created Before Release 15.0

When retrieving assemblies created prior to Release 15.0, you can use the **ReIntersect** command in the ASSY FEAT menu to update old style assembly features to the Release 15.0 style assembly features and remove unnecessary external references. The **ReIntersect** command is available in the ASSY FEAT menu only if you have set the configuration file option `allow_redo_intersections` to `yes`.

A remnant is geometry that an assembly feature creates that is not in session. A possible cause of remnants is that assembly features are visible in the assemblies being reintersected, but these features were created in a higher-level assembly or one that is in a different assembly tree. Another possible cause is that part or assembly files have been manipulated in the operating system without using Pro/ENGINEER or Pro/PDM. You can remove remnant geometry only if you have set the configuration file option `fix_refs_to_intersections` to `yes` (no is the default value).

## About Restructuring an Assembly

Using the **Restructure** command in the ASSEMBLY menu, you can modify assembly groupings and add flexibility to the assembly design process. You can move components from one subassembly to another, or from the top-level assembly to a subassembly, or vice versa. However, the top-level assembly must be in session to properly regenerate a restructured subassembly. Components may have to be frozen otherwise. After you restructure a subassembly, you can add to it and modify it.

## To Move a Component to a Different Level

When you choose **Restructure**, the Model Tree window displays the assembly hierarchical tree structure, enabling you to restructure it without changing the appearance of the assembly in the graphics window.

Use commands from the Model Tree pull-down menus to highlight a selected assembly member in the graphics window or change the view of the assembly and the displayed level of structure.

1. Choose **Restructure** from the ASSEMBLY menu. The RESTRUCTURE menu appears.
2. Choose **Move Comp** and select a component or subassembly to move. The system tags the selected member as MOVING in the Restructure Status column of the Model Tree.
3. Choose **Select Target**. In the Model Tree window, select an assembly to which you want to move the selected component. The selected component moves to the designated assembly.  
You can choose **Undo Last** to undo the last move.
4. Choose **Done Sel**, and **Done** from the RESTRUCTURE menu.

## Restructuring Assembly Components

You can use **Restructure** if you have an assembly with elements, and you want to handle the conglomerate assembly as one entity. You can create an empty subassembly in the assembly, and then restructure all the elements into that subassembly. You can then create a drawing of the subassembly and list the components in a table for a BOM. However, the first time you retrieve the subassembly without the top-level assembly, you must freeze all the components.

The following restrictions apply to restructuring assembly components:

- You cannot select the first member of an assembly to restructure.
- When restructuring, if both the original component and the target assembly are members of the same level assembly, you must move the children of the original component as well.
- You cannot restructure components that are part of a pattern.
- If an assembly contains multiple copies of the same subassembly, you cannot restructure the components of that subassembly.
- The system does not allow you to restructure an assembly if multiple occurrences of the original assembly exist.

## About Regenerating Parts Modified in Assembly Mode

You need to use the **Regenerate** command only on the parts that you have modified while in Assembly mode.

In large assemblies, it is faster to select individual parts to regenerate than it is to instruct Pro/ENGINEER to search for and regenerate all parts in the assembly that you have modified.

When you modify a part intersected by an assembly feature in an assembly in another Pro/ENGINEER session, you must regenerate the part in the assembly before the system makes the changes visible.

Before regenerating a part with external references, the system regenerates the external reference and then uses the updated values for the part.

## To Update Placement Without Regenerating

If you do not select any parts when regenerating, the system regenerates only assembly placement constraints and datum features.

1. Choose **Regenerate** from the ASSEMBLY menu. The PRT TO REGEN menu appears.
2. Choose **Select**, and choose **Done** from the SELECT PARTS menu instead of **Pick Part** or **Layer** if you only want to update component placement without regenerating parts.
3. If assembly features are present, choose **Dont Update** and **Done** from the REG INT PRT menu.

## To Regenerate Selected Parts

1. Choose **Regenerate** from the ASSEMBLY menu. The PRT TO REGEN menu appears.
2. Choose **Select**. The SELECT PARTS menu appears.
3. Do one of the following:
  - Choose **Pick Part** and select individual parts in the graphics window, or select them by menu.
  - Choose **Layer** and select parts by specifying the name of a layer that contains them.

If assembly features are present, choose one of the following from the REG INT PRT menu:

- **Upd Int Prts**—Updates all parts intersected by assembly features.
  - **Dont Update**—Does not update any part intersected by assembly features.
4. Choose **Done**, and then **Done** from the SELECT PARTS menu to regenerate the selected parts, or choose **Quit**, and then **Quit Regen** from the PRT TO REGEN menu to abort the process.

## To Regenerate All Changed Parts

1. Choose **Regenerate** from the ASSEMBLY menu. The PRT TO REGEN menu appears.
2. Choose **Automatic** to regenerate every part that has changed since the last regeneration. Pro/ENGINEER

selects models with external references to parts that have changed.

## To Customize Regeneration

1. Choose **Regenerate** from the ASSEMBLY menu. The PRT TO REGEN menu appears.
2. Choose **Custom**. The Regeneration Manager appears.  
Initially, the system expands the tree to the first level in the Regeneration list column.
  - To display features, choose **Show** from the menu bar.
  - To expand or collapse the tree to any level, choose **View** from the menu bar.Use the **Info** menu to obtain information about features and components.
3. From the list, select components and features for regeneration; then do one of the following:
  - To select all components for regeneration, select **Regenerate** and then **Select All**.
  - To omit all components from regeneration, select **Skip Regen** and then **Select All**.
  - To determine the reason an object requires regeneration, select **Highlight** and select an object in the tree.If you select a subassembly, the system selects all parts and features within that subassembly even if it is broken up in the regeneration list. A blank column next to the regeneration list indicates whether you have selected a component or feature for regeneration.
4. Click **OK** to close the dialog box and regenerate the components you have selected.

## About Resolving Failures

When a component fails during regeneration or retrieval, the error messages include the component identification number as well as the model name.

A component may be missing for these reasons:

- It was misplaced in the directory tree.
- It was renamed.
- It was deleted from the disk.

If a component is missing upon retrieval, Pro/ENGINEER assembles all parts until it requires the missing part; then it displays the RESOLVE FEAT menu. The system temporarily removes all later components until it recovers the missing component. After the system replaces or removes the component, it returns all the temporarily removed members to their places.

After you place a component, you may not be able to assemble it to other components because of one of the following problems:

- A missing referenced feature
- A violated placement constraint

## To Resolve a Missing Component Problem

When a component cannot be retrieved because it is missing, you can do the following:

- Click **Quick Fix > Find Component** to open the **File Open** browser. Browse your directories to locate the missing component. Select and open the missing component. You can locate multiple missing components in this way. Retrieval of the assembly should continue normally.
- Click **Quick Fix > Quit Retr** or **Fix Model > Quit Retr**. **Quit Retr** erases from memory all objects that are retrieved from disk during the retrieval. Use the **Quit Retr** command to quit the retrieval so that you can begin again. Objects in session before the retrieval of the top object is initiated remain in session. You can then go back and rename the part or correct the pathname so the system can find it. You must then retrieve the assembly again.

- Click **Fix Model > Component > Adv Utils > Replace**. At the system prompt, enter the name of the component. If you enter [?], the system finds the component for you. The REPLACE WITH menu appears with the **Manual** command highlighted. Assemble the new component. Reconstruction of the assembly should then continue.
- Use **QUICK FIX > Suppress**, **QUICK FIX > Redefine**, or **QUICK FIX > Delete** to suppress, redefine, or delete the failed component. These commands only apply to the failed component. To suppress, redefine, or delete another component in the assembly, use the commands in the FIX MODEL menu.
- Use commands in the FIX MODEL menu, which is a subset of the ASSEMBLY menu.

**Note:** After recovering from an error, save the new version of the assembly before exiting from Pro/ENGINEER.

## To Recover a Failed Assembly with a Renamed Component

If you rename a component on disk but do not save the assembly before you exit, reconstruction of the assembly in the next session fails and the message `Cannot retrieve` appears in the message window. In this case, you must retrieve the renamed component and reconstruct the assembly:

1. Choose **Fix Model > Component > Adv Utils > Replace > Failed Feat**. The Open dialog box appears.
2. Select the new name for the component.
3. Specify the placement constraints and assemble the component using the Component Placement dialog box to complete the assembly reconstruction manually.

**Note:** Save the assembly with the new name of the component before exiting from Pro/ENGINEER.

## To Resolve a Component Placement Problem

When a component cannot be placed because references are missing, the system displays an error message, the RESOLVE FEAT menu, and the Failed Feature Diagnostic window.

When the system cannot place a component, you can do the following:

- Click **Quick Fix > Find Component** to open the **File Open** browser. Browse your directories to locate the missing component. Select and open the missing component. You can locate multiple missing components in this way.
- Click **Quick Fix > Freeze**. The system then places the component (nonparametrically) in its last known placement. You can redefine it or change it later.
- Click **Quick Fix > Redefine**. Use the Model Tree window to redefine the location of the component.
- Click **Quick Fix > Suppress** or **Quick Fix > Delete** to suppress or delete the failed component.
- Use commands in the FIX MODEL menu, which is a subset of the ASSEMBLY menu.

**Note:** The commands in the QUICK FIX menu apply only to the failed component. To suppress, redefine, or delete another component in the assembly, use the commands in the FIX MODEL menu.

**Note:** After recovering from an error, save the new version of the assembly before exiting from Pro/ENGINEER.

## To Resolve Failure to Retrieve an Assembly Feature

When retrieving an assembly, you may not be able to regenerate a feature such as a datum or hole because the part or a part feature that it references is deleted, or the part feature is suppressed.

Whenever the system cannot place datums, they appear in the locations of the last successful regeneration. If a component references any of these datums, Pro/ENGINEER issues an error message and displays the RESOLVE

FEAT menu. You can specify its new placement, or choose **Quick Fix**, and then **Freeze** to place it nonparametrically.

If the datum references were suppressed, you can unfreeze the datums by resuming them. Successful regeneration occurs, and the components placed using the **Freeze** command also regain parametric placement.

**Note:** After recovering from an error, save the new version of the assembly before exiting from Pro/ENGINEER.



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